

EngiLab Truss.2D 2022
v1.3

User Manual

EngiLab

www.engilab.com

This page intentionally left blank.

EngiLab Truss.2D 2022 v1.3 User Manual

(c) 2022 EngiLab PC

All rights reserved. No parts of this work may be reproduced in any form or by any means - graphic, electronic, or mechanical, including photocopying, recording, taping, or information storage and retrieval systems - without the written permission of the publisher.

Products that are referred to in this document may be either trademarks and/or registered trademarks of the respective owners. The publisher and the author make no claim to these trademarks.

While every precaution has been taken in the preparation of this document, the publisher and the author assume no responsibility for errors or omissions, or for damages resulting from the use of information contained in this document or from the use of programs and source code that may accompany it. In no event shall the publisher and the author be liable for any loss of profit or any other commercial damage caused or alleged to have been caused directly or indirectly by this document.

Publisher

EngiLab PC

Document type

User Manual

Program name

EngiLab Truss.2D

Program version

v1.3.8183.0

Document version

v1.0

Document release date

May 28, 2022

"Perfection is achieved, not when there is nothing more to add, but when there is nothing left to take away."

Antoine de Saint-Exupéry

EngiLab

This page intentionally left blank.

Table of Contents

Chapter 1 Introduction to EngiLab Truss.2D 1

1 Overview 2
2 What is a 2D Truss? 3
3 Ready to-analyze Examples 5
4 Lite and Pro Editions 7

Chapter 2 Structural Modeling 11

1 Setting up a model 12
2 Model Properties 15
Units 15
Consistent System 17
Predefined Systems 23
Custom System 25
Materials 26
Sections 30
Import Sections from Database 37
Nodes 38
Elements 53
Nodal Loads 63
Elemental Loads 66
Body (Acceleration) Loads 71

Chapter 3 Tools 75

1 Import Materials/Sections 76
2 Import DXF 77

Chapter 4 Analysis, Results and Validation 79

1 Analysis 80
2 Axial Diagrams 83
3 Color Plots 86
4 Deformation 89
5 Free Body Diagram 92
6 Analysis Results Form 94
Node Displacements 94
Support Reactions 95
Element Results 96
Element Checks 98
Full Report 100
7 Analysis Validation Form 102
Global (Model) Equilibrium 103
Energy Conservation 104
Joints (Nodes) Equilibrium 106
Elements Equilibrium 107

(c) 2022 EngiLab PC

Chapter 5 Settings 109

1 General Settings ... 110
2 Model Settings ... 111
3 Axial Diagrams/Color Plots Settings ... 113
4 Colors Settings ... 115
5 Fonts Settings ... 118
6 Report Settings ... 119

Chapter 6 Example Problems 121

1 Example Problem 1 ... 122
Example 1 - Hand Calculations ... 123
Step 1. Define the System of Units ... 125
Step 2. Define Materials ... 125
Step 3. Define Sections ... 127
Step 4. Draw the Model on screen ... 129
Step 5. Edit Nodes ... 138
Step 6. Edit Elements ... 141
Step 7. Define Nodal Loads ... 142
Step 8. Define Elemental Loads ... 147
Step 9. Define Body (Acceleration) Loads ... 148
Step 10. Run the Analysis ... 150
Step 11. View the Analysis Results on screen ... 150
Step 12. View the analytical results ... 162
Analysis Validation (optional) ... 167

Chapter 7 License Agreement 171

1 EULA (Lite Edition) ... 172
2 EULA (Pro Edition) ... 174

(c) 2022 EngiLab PC

Chapter



Introduction to EngiLab Truss.2D

1 Introduction to EngiLab Truss.2D

1.1 Overview



EngiLab Truss.2D 2022

v1.3 (v1.3.8183.0)

Copyright (c) 2022 EngiLab PC.
All rights reserved



www.engilab.com

EngiLab Truss.2D is a powerful structural engineering software for the **linear static analysis of plane (2D) trusses** for Windows, featuring a fast, rich and intuitive user interface for pre-processing and post-processing of the results. It uses the **Finite Element Method (FEM)** for the analysis, with 2 DOFs per Node. EngiLab Truss.2D can draw the Model Deformation, Free Body Diagram, Axial Force/Stress/Strain Diagrams or Color Plots, among others.

Communication

For general inquiries, please contact us at info@engilab.com

Technical support

EngiLab offers technical support via email at support@engilab.com. Email questions are normally answered within 24 hours for registered users. Considering our customer support team's busy work schedule, EngiLab strives to provide technical support, via email, to non-registered (Lite Edition) users as well.

Required Information for Support

When sending us a question via email, please make sure to include the following information:

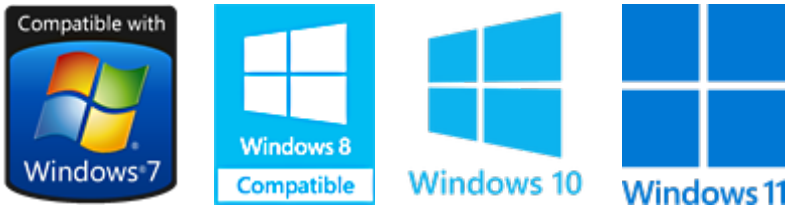
- Operating system (Windows 7, 8, 10, 11, etc.)
- Name and exact version of the product being used.
- Brief description of the problem.
- Detailed instructions on how to reproduce the problem, and/or an input file as an attachment.

You may find additional information, new releases, etc on the web at www.engilab.com. We take customer satisfaction very seriously and always welcome your feedback.

Compatibility

EngiLab Truss.2D is compatible with the following versions of Windows (32 bit and 64 bit):

- Windows Vista, Windows 7, Windows 8, Windows 8.1, Windows 10, Windows 11
- Windows Server 2003, Windows Server 2008, Windows Server 2008 R2, Windows Server 2012



Limited Warranty - Disclaimer of Damages

See the License Agreement.

1.2 What is a 2D Truss?

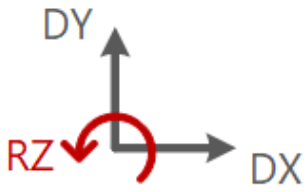
About 2D Trusses

EngiLab Truss.2D specializes in the analysis of 2D Trusses. In EngiLab Truss.2D, each Node has 2 Degrees Of Freedom (DOFs), the X- and the Y- displacement. There is no rotational DOF, as all members are free to rotate at their ends, unlike in 2D frames. The figure below explains this and highlights the differences between **EngiLab Truss.2D** and **EngiLab Frame.2D** programs.

(c) 2022 EngiLab PC



2D Truss
2 DOFs per Node



2D Frame
3 DOFs per Node



EngiLab Truss.2D



EngiLab Frame.2D



Axial Force only



Axial Force
Shear Force
Bending Moment

Trusses are used commonly in Steel buildings and bridges. A Truss has the following properties:

- All the members are straight and remain straight after the deformation.
- Members are connected together with pin joints.
- Members are connected only at the ends of the members.
- **All external forces (loads & reactions) must be applied only at the joints.**
- As a result, the members have only axial forces.
- Every member of a truss is essentially a 2 force member (see above).
- There is no shear and no bending within the Element.

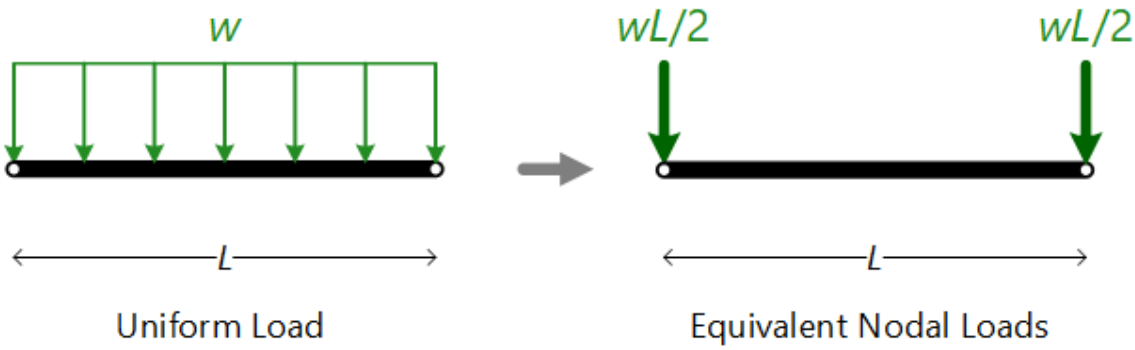
Elemental (Uniform) Loads in 2D Trusses

Trusses are usually assumed to be of negligible weight (compared to the loads they carry). If self-weight of the members or other distributed loads (such as wind, snow and inertial forces) need to be taken into account, then they must be converted to equivalent Nodal Loads. In the end, for the Analysis needs, there will be only Nodal Loads on the structure. Doing this, the effects of bending on the Elements are neglected

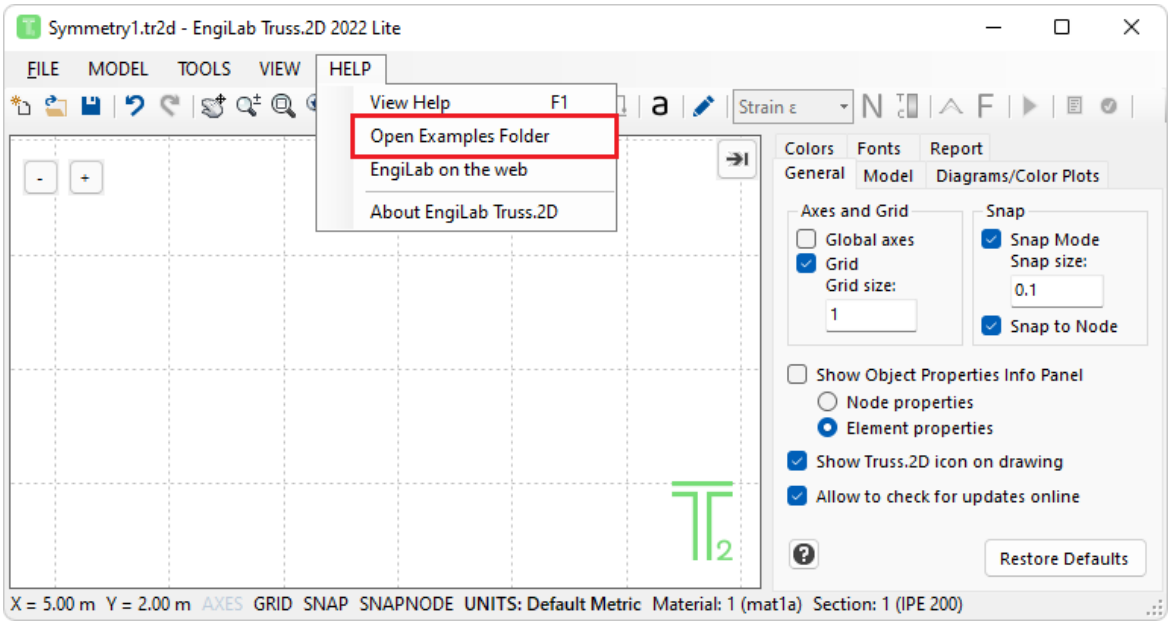
(c) 2022 EngiLab PC

This is the approach that EngiLab Truss.2D also uses. Self weight can be applied, as discussed in the Body (Acceleration) Loads section, yet for the analysis it is automatically converted to equivalent Nodal Loads. The same is true for Elemental (Uniform) Loads defined on any member. Uniform loads can be applied to members, but again they are automatically converted to equivalent Nodal Loads before the Analysis takes place. As a result, the Model to be analyzed has all the properties of a 2D Truss Model, discussed above.

Assuming that a 2D Truss member carries a uniform load w (for example, in kN/m units), the total load is F=w*L (in kN), where L (in m, in this example) is the Length of the Element. For the Analysis, each Node (start and end Node) of this members takes a Force equal to F/2=w*L/2. This is shown schematically below.



1.3 Ready to-analyze Examples



(c) 2022 EngiLab PC

In the folder **\Examples** (which is located where the program was installed), you can find ready-to-analyze EngiLab Truss.2D input files (.tr2d) that you can open and analyze within EngiLab Truss.2D.

- There are 8 example files in total

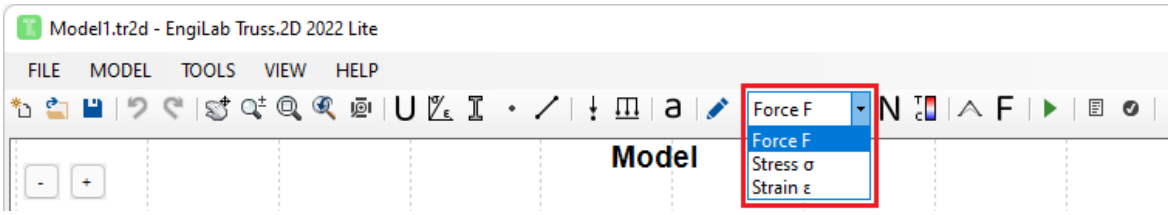
How to open and Analyze an example:

- From the FILE menu, select Open.
- Open the Examples folder (Usually **C:\Program Files (x86)\EngiLab\EngiLab Truss.2D 2021 Lite\Examples** for the Lite Edition and **C:\Program Files (x86) \EngiLab\EngiLab Truss.2D 2021 Pro\Examples** for the Pro Edition.
- Select a .tr2d file to open.
- After the file is opened, click the  (Analyze) button to analyze the model.

(c) 2022 EngiLab PC

After the analysis is finished:

Click N to see the **Axial Diagrams**, by making a selection on the Dropdown menu



- **Axial Force** Diagram
- **Axial Stress** Diagram
- **Axial Strain** Diagram

Click to see the **Color Plots**:

- **Axial Force** Color Plot
- **Axial Stress** Color Plot
- **Axial Strain** Color Plot

Click to see the deformed shape of the Model.

- : Deformation

Click F to see the Free Body Diagram of the Model.

- F: Free Body Diagram

Click to see the analytical results. The results include the following tabs:

- Node Displacements
- Support Reactions
- Element Results
- Element Checks
- Full Report

1.4 Lite and Pro Editions

EngiLab Truss.2D comes in two Editions: **Lite** and **Pro**. The Lite Edition is completely free, but it has some limitations compared to the full Pro Edition, as described in detail below.

Lite Edition

- **Completely free of charge**, available for download at www.engilab.com

(c) 2022 EngiLab PC

- Only for **non-profit use** (personal, academic or educational purposes). It may *not* be used for any commercial purposes.
- **Analysis limitations**: The Lite Edition can open, save and modify any structural Model. Its input files are exactly the same as the ones of the Pro Edition. Yet, the Lite Edition can only analyze small Models of up to **10 Nodes, 3 Materials and 3 Sections**.
- **No missing features**: You get the real experience of the Pro Edition. All the functions of the Pro Edition are present also in the Lite Edition. No trial version banners printed anywhere, no annoying pop-ups, no hidden fees, no surprises.
- Ideal for: Students and Hobbyists who need to analyze small models, or Professional Engineers who would like to test the Program before purchasing the Pro Edition.
- Technical Support for the Lite Edition is only provided via email at support@engilab.com

(c) 2022 EngiLab PC

Pro Edition

- The Pro Edition is available for purchase at www.engilab.com.
- It can be used for **any purpose** (personal, academic, educational or **commercial use**)
- **No modeling limitations**: The Pro Edition can analyze any structural model (Unlimited number of Nodes, Materials and Sections).
- Ideal for: Professional Engineers, Universities, Students, Researchers, Hobbyists who wish to analyze models without any limitations.
- **Priority Technical Support** is provided within 24 hours via email at support@engilab.com, or by phone.

(c) 2022 EngiLab PC

This page intentionally left blank.

Chapter



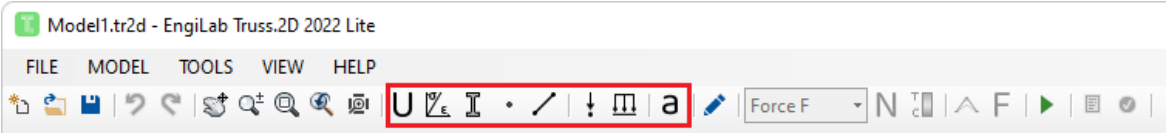
Structural Modeling

2 Structural Modeling

2.1 Setting up a model

The user can set-up, analyze and view the results of a model using the following **12 simple steps**:

A. Pre-Processing: Building the Model



1. Click U to define the **System of Units**. This is a very important step for the model. For details, see Units

2. Click to define **Material**(s). You need to define the Material **Name** (optional), **Elastic Modulus E**, **Density d** (optional) and **Yield Stress fy** (optional) for each Material. For details, see Materials.

3. Click I to define **Sections**(s). You need to define the Section **Name** (optional), **Area A** and **Moment of Inertia I** (optional) for each Section. For details, see Sections.

4. After you have defined at least one Material and one Section, you can **start drawing your Model on screen**, as follows:

- Left-click on screen to define a new free Node (with no constraints).
- Left-click on screen, hold down the left button and then release it at another location to define a new Element and two nodes at ends i, j.
- Double-click on a Node or close to it to to define a new Constraint (Support) set. Each time you double-click a Node, a new Nodal Constraint set is applied to the Node.
- If **SNAP** is activated (Default=True), then you can only draw Nodes and Elements at increments defined by the Snap Size setting (Default=0.1).
- If **SNAPNODE** is activated (Default=True), then you can "catch" Nodes so that new elements can be connected to existing Nodes.
- **Note**: All Elements that are defined on screen are assigned the Default Material and the Default Section. The Default Material and Section can be selected in the Materials and Sections forms, respectively.

(c) 2022 EngiLab PC

If you do not want to use on-screen drawing, you can still click to define Nodes manually and to define Elements manually.

5. Click to:

- Move existing **Nodes** to their exact positions, if needed. For example, a Node with X-Coordinate 5.8 is defined on screen with Snap Size = 0.1, but it should be moved to the exact position 5.85.
- Define or change Nodal Constraints (supports).
- Define Springs.
- Use tools related to Nodes, such as the Move Nodes tool, Copy Nodes tool, and others.

For details, see Nodes.

6. Click to:

- Assign the right Material and Section to every **Element**, if needed. All Elements that are defined on screen are assigned the Default Material and the Default Section, but the user can change this later on, if needed.
- See properties of Elements such as Length, and others.

For details, see Elements.

7. Click to define **Nodal Loads**. For details, see Nodal Loads.

8. Click to define **Elemental Loads**. For details, see Elemental Loads.

9. Click to define **Body (Acceleration) Loads** (if needed), for example to take into account the self weight of the structure. For details, see Body (Acceleration) Loads.

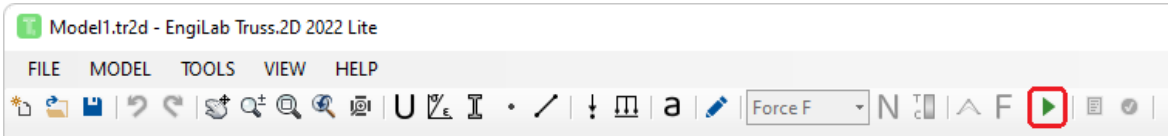
- If you want to take into account the self-weight of Elements as an additional uniform load for each Element, then you have to provide the Material Density for the Material of each Element and also to define a Linear Acceleration Vector equal to the standard earth gravitational acceleration. A common practice is to set the earth gravitational acceleration with a minus (-) sign at the Y direction - this means gravity acting towards -Y global axis.
- **Example:**

(c) 2022 EngiLab PC

Using Consistent Units, if you are using kN for forces, m for length and s for time, then the Material Density has to be given in t/m^3 and you have to enter -9.80665 (or simply -9.81) at the aY component of the Linear Acceleration Vector.

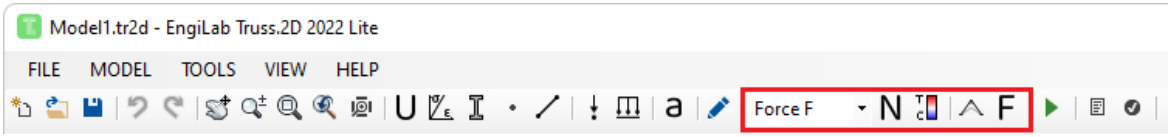
Using another System of Units, you simply have to enter the value of the standard earth gravitational acceleration in this system (check the Acceleration unit used in Units).

B. Analyze the Model



10. Click (or press F5) to **run the Finite Element Analysis**. For details on the Analysis, see Analysis.

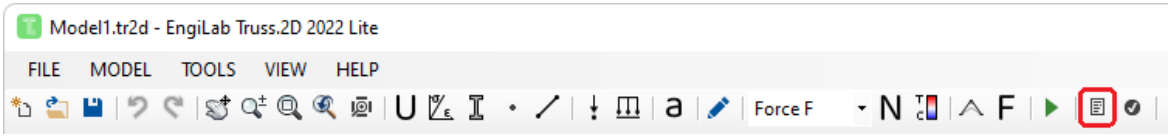
C. Post-Processing: See the Analysis Results



11. Click N, , or F to see the **Axial Diagrams**, **Color Plots**, **Model Deformation** or the **Free Body Diagram**, respectively.

- N: Axial Diagrams for Force, Stress or Strain
- : Color Plots for Force, Stress or Strain
- : Deformation (deformed shape of the Model)
- F : Free Body Diagram (external forces together with the support reactions)

12. Click to see the **analytical results**.

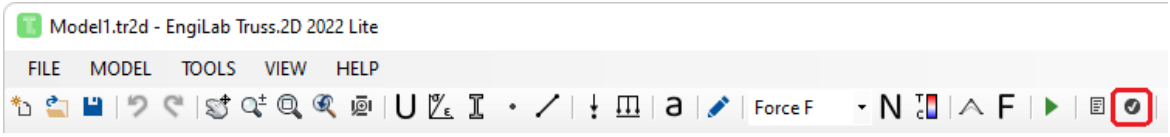


The analytical results include:

- Node Displacements
- Support Reactions
- Element Results

- Element Checks
- Full Analysis Report

D. Analysis Validation (optional)

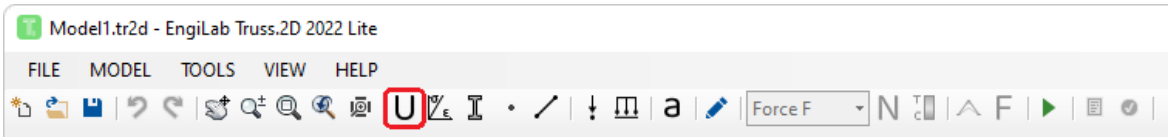


Optionally, you can click to see the Analysis Validation Reports and make sure that everything is OK with the Analysis and its results.

- Global (Model) Equilibrium
- Energy Conservation
- Joints (Nodes) Equilibrium
- Elements Equilibrium

2.2 Model Properties

2.2.1 Units



Before starting to define any model in the program, you need to select a **System of Units** to use. EngiLab Truss.2D supports various Systems of Units:

- **Consistent System**
- **Predefined Systems**
  - Default Metric
  - Default US
  - International SI
- **Custom System (user-defined)**

EngiLab strongly recommends the use of "EngiLab Units" software for any unit conversions needed. EngiLab Units is a free but powerful unit conversion program that is available for download at www.engilab.com. EngiLab Units 2022 (v3.2) supports 645 units in 21 unit categories, including distance, acceleration, pressure (stress) and others. **All units needed in EngiLab Truss.2D are supported by EngiLab Units**.

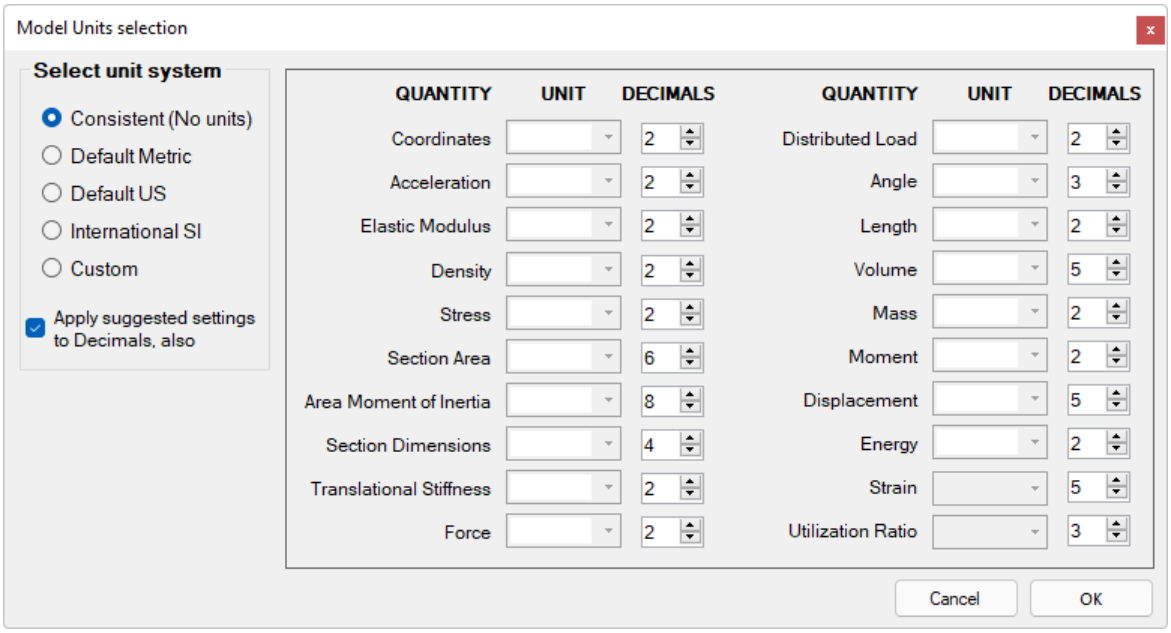
(c) 2022 EngiLab PC



The Pre-defined Systems of Units are standard and cannot be changed. They are based on commonly used unit systems around the world.

The Custom System is fully customizable by the user.

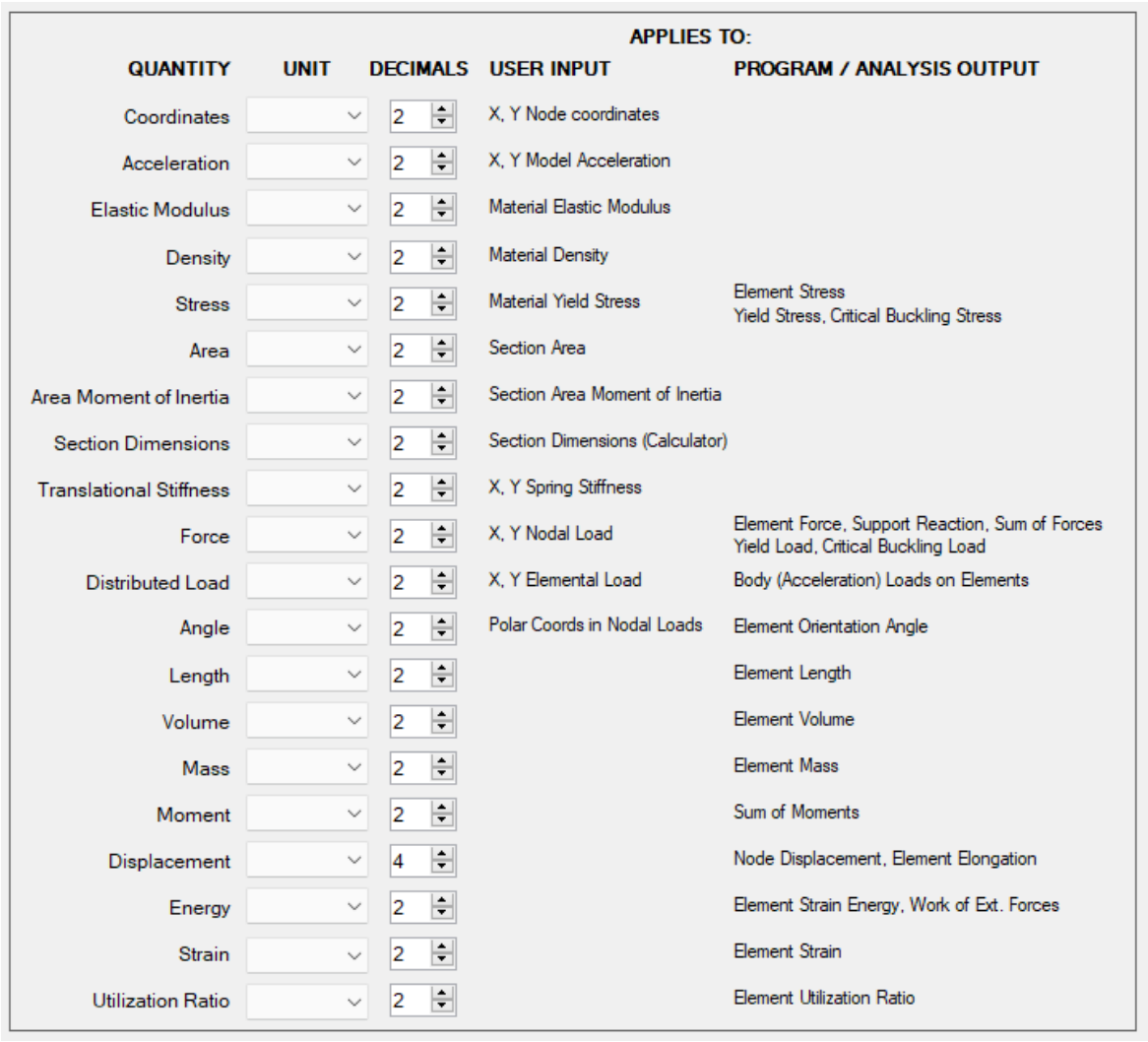
If you click the "Units" button in the Toolbar, you get the Units form. The screenshot below depicts the Consistent System of Units. This system is special and is explained in detail in the Section Consistent Units



Consistent System of Units

The pre-defined Systems of Units are standard and cannot be changed. The Custom system is fully customizable by the user.

There are **20 types of units** used in the program. The various Units and their use, are explained in detail in the following figure:



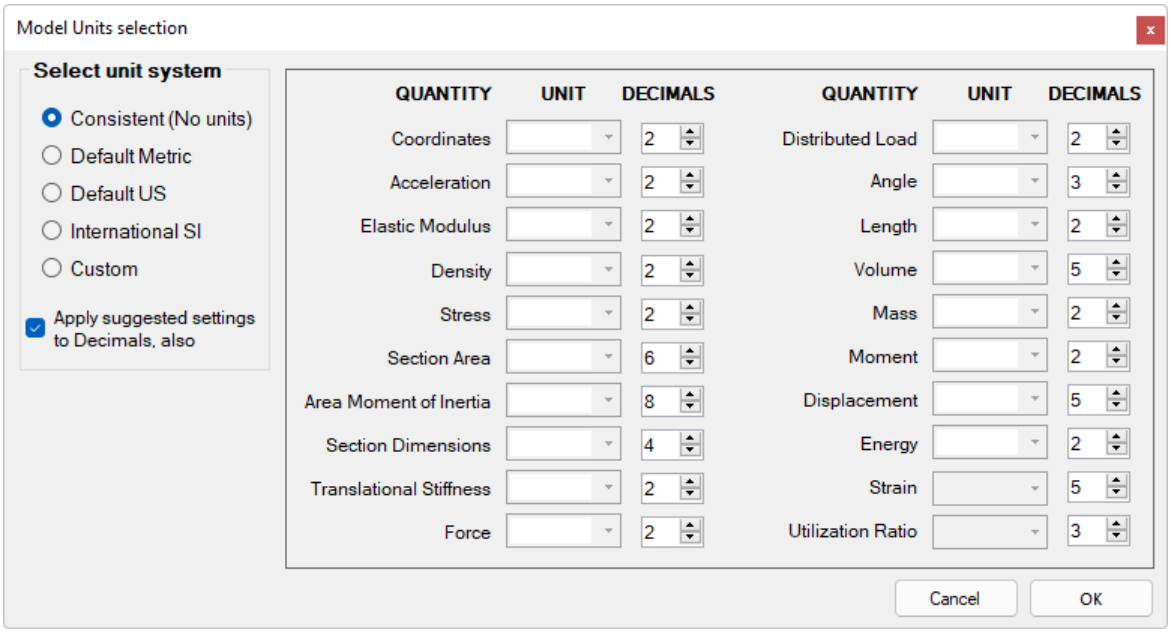
			APPLIES TO:	
QUANTITY	UNIT	DECIMALS	USER INPUT	PROGRAM / ANALYSIS OUTPUT
Coordinates		2	X, Y Node coordinates	
Acceleration		2	X, Y Model Acceleration	
Elastic Modulus		2	Material Elastic Modulus	
Density		2	Material Density	
Stress		2	Material Yield Stress	Element Stress Yield Stress, Critical Buckling Stress
Area		2	Section Area	
Area Moment of Inertia		2	Section Area Moment of Inertia	
Section Dimensions		2	Section Dimensions (Calculator)	
Translational Stiffness		2	X, Y Spring Stiffness	
Force		2	X, Y Nodal Load	Element Force, Support Reaction, Sum of Forces Yield Load, Critical Buckling Load
Distributed Load		2	X, Y Elemental Load	Body (Acceleration) Loads on Elements
Angle		2	Polar Coords in Nodal Loads	Element Orientation Angle
Length		2		Element Length
Volume		2		Element Volume
Mass		2		Element Mass
Moment		2		Sum of Moments
Displacement		4		Node Displacement, Element Elongation
Energy		2		Element Strain Energy, Work of Ext. Forces
Strain		2		Element Strain
Utilization Ratio		2		Element Utilization Ratio

Note that some units are used only for Model input data (for example, the Elastic Modulus unit), while other units are used for both inputs and outputs (for example, the Force unit or the Stress unit). Others are used only for outputs (for example, the Displacement unit, or the Moment unit).

2.2.1.1 Consistent System

The Consistent System of Units is the one shown in the form below. In fact, in this system, no specific units are used for any of the quantities, this is why all unit fields are empty. The Units can be anything (any compatible unit for the quantity in question), provided that some rules are followed, as explained next.

(c) 2022 EngiLab PC



Consistent System of Units

How does a Consistent System of Units work?

Imagine Newton's Second Law of Motion. The equation is ***F = m * a***,

where *F* is the force, *m* is the mass of the object and *a* is the acceleration. The second law states that the rate of change of momentum of a body over time is directly proportional to the force applied, and occurs in the same direction as the applied force. This law is global and it applies everywhere, no matter what units we use for the three quantities (Force, Mass and Acceleration). But these units need to be consistent, i.e. the units we use for mass and acceleration will define the unit we take for force when we apply the formula (without any conversions). For example if we use m=20 kg and a=10 m/s^2, then we take F=200 and the unit of Force is then kg*m/s^2, which means N (Newton). The result is therefore F=200 N. In a similar way, we can use the same formula with different units, as long as we always keep things consistent.

As long as we use **Consistent Units**, there is no need for any unnecessary conversions. What you give (inputs) will define what you take (outputs). When using Consistent Units, the program will not show any units, but the user needs to know what units he/she uses for the inputs. The outputs will simply comply. Below you can find a thorough explanation of how exactly this works and also instructions on using the most common units this way.

Implementation

All input data must be specified in **consistent units**. As a result, the analysis results will also comply to that system. The important point about using Consistent Units is the necessity to stick with units that work correctly together - not to mix units that do not have a correct relationship with each other.

In order to define a consistent system of units, you have to define first the **primary (basic) units** and then the **derived units** which are dependent on the primary units. We propose two different approaches for defining a consistent system of units, as described in detail below:

(c) 2022 EngiLab PC

A. Consistent system of units based on Force

Define the three primary (basic) units for **Force (F), Length (L), Time (T)**. For example you can choose to use kN, m, s (sec). The **derived units** are then the following:

Derived Unit	Formula	Formula explanation	In our example (kN, m, s)
Acceleration	L/T^2	(1 Length unit) / (1 time unit)^2	m/s^2
Mass *	$F{\cdot}T^2/L$	(1 force unit) / (1 acceleration unit)	$kN/(m/s^2)$ = Mg = t *
Density	$F{\cdot}T^2/L^4$	(1 mass unit) / (1 length unit)^3	t/m^3
Stress	F/L^2	(1 force unit) / (1 length unit)^2	kN/m^2

***** The mass unit (in our example 1 t) is the mass that accelerates by the acceleration unit rate (in our example 1 m/s^2) when the unit force (in our example 1 kN) is exerted on it.

B. Consistent system of units based on Mass

Define the three primary (basic) units for **Mass (M), Length (L), Time (T)**. For example you can choose to use kg, m, s (sec). The **derived units** are then the following:

Derived Unit	Formula	Formula explanation	In our example (kg, m, s)
Acceleration	L/T^2	(1 Length unit) / (1 time unit)^2	m/s^2
Force *	$M{\cdot}L/T^2$	(1 mass unit) · (1 acceleration unit)	$kg{\cdot}m/s^2$ = N *
Density	M/L^3	(1 mass unit) / (1 length unit)^3	kg/m^3
Stress	$M/L/T^2$	(1 force unit) / (1 length unit)^2	N/m^2

***** The force unit (in our example 1 N) is the force required to accelerate the unit mass (in our example 1 kg) at the acceleration unit rate (in our example 1 m/s^2).

(c) 2022 EngiLab PC

Common consistent systems of units

Some common systems of consistent units are shown in the table below. Any column can be used for the inputs (and outputs) of the program.

Quantity	SI (MKS)	MTS	mmNS	US Unit (ft)	US Unit (in)
Length	m	m	mm	ft	in
Force	N	kN	N	lbf	lbf
Mass	kg = N·s^2/m	t (tonne) = kN·s^2/m	t = N·s^2/mm	slug = lbf·s^2/ft	blob = lbf·s^2/in
Time	s	s	s	s	s
Stress	Pa (N/m^2)	kPa (kN/m^2)	MPa (N/mm^2 = MN/m^2)	lbf/ft^2	psi (lbf/in^2)
Density	kg/m^3	t/m^3	t/mm^3	slug/ft^3	lbf·s^2/in^4
Acceleration	m/s^2	m/s^2	mm/s^2	ft/s^2	in/s^2

As points of reference, the mass density of steel, the Young's Modulus of steel and the standard earth gravitational acceleration are given for each system in the table below.

Quantity	SI (MKS)	MTS	mmNS	US Unit (ft)	US Unit (in)
Steel density	7850 kg/m^3	7.85 t/m^3	7.85×10^{-9} t/mm^3	15.23151461 slug/ft^3 (lbf·s^2/ft^4)	7.34544493×10^{-4} blob/in^3 (lbf·s^2/in^4)
Steel Modulus of Elasticity (E)	210×10^9 N/m^2	210×10^6 kN/m^2	210×10^3 MPa	4,385,941,188.96153 lbf/ft^2 (psf)	30,457,924.9233 lbf/ft^2 (psi)
Earth Gravity acceleration	9.80665 m/s^2	9.80665 m/s^2	9806.65 mm/s^2	32.17404856 ft/s^2	386.088583 in/s^2

(c) 2022 EngiLab PC

Notes:

- 1 t (tonne) = 10^3 kg = 1 Mg. It is a mass that accelerates by 1 m/s^2 when a force of 1 kN is exerted on it.
- 1 slug = 1 lbf·s^2/ft. It is a mass that accelerates by 1 ft/s^2 when a force of 1 pound-force (lbf) is exerted on it.
- 1 blob = 1 lbf·s^2/in. It is a mass that accelerates by 1 in/s^2 when a force of 1 pound-force (lbf) is exerted on it.
- 1 MPa = 1 MN/m^2 = 1 N/mm^2

Practical example

The user chooses to use the **MTS system**, a common choice for structural engineering applications:

- Length: **m**
- Force: **kN**
- Time: **s**

(c) 2022 EngiLab PC

EngiLab Truss.2D data have to be given as shown below:

Quantity	**Unit used**
Node Coordinates X, Y	m
Material Elastic Modulus E	kPa = kN/m^2
Material Density d	t/m^3
Section Area A	m^2
Section Moment of Inertia I	m^4
Nodal Force F	kN
Elemental load f	kN/m
Spring Elastic Stiffness KX, KY	kN/m
Acceleration	m/s^2

The results will also comply to that system, thus they will be given as:

Quantity	**Unit used**
Node X, Y Displacement	m
Axial Force	kN
Axial Stress	kPa
Support reaction FX, FY	kN
Strain Energy	kN·m=kJ
Mass	t=Mg
Volume	m^3

* Rotations (where applicable) are ALWAYS given in RADIANS in a Consistent System of Units.

Note: In the above example, if one wants to apply self-weight to the structure, he/she can add the standard earth gravitational acceleration at the -Y direction as: **aY = -9.80665** (i.e. m/s^2)

2.2.1.2 Predefined Systems

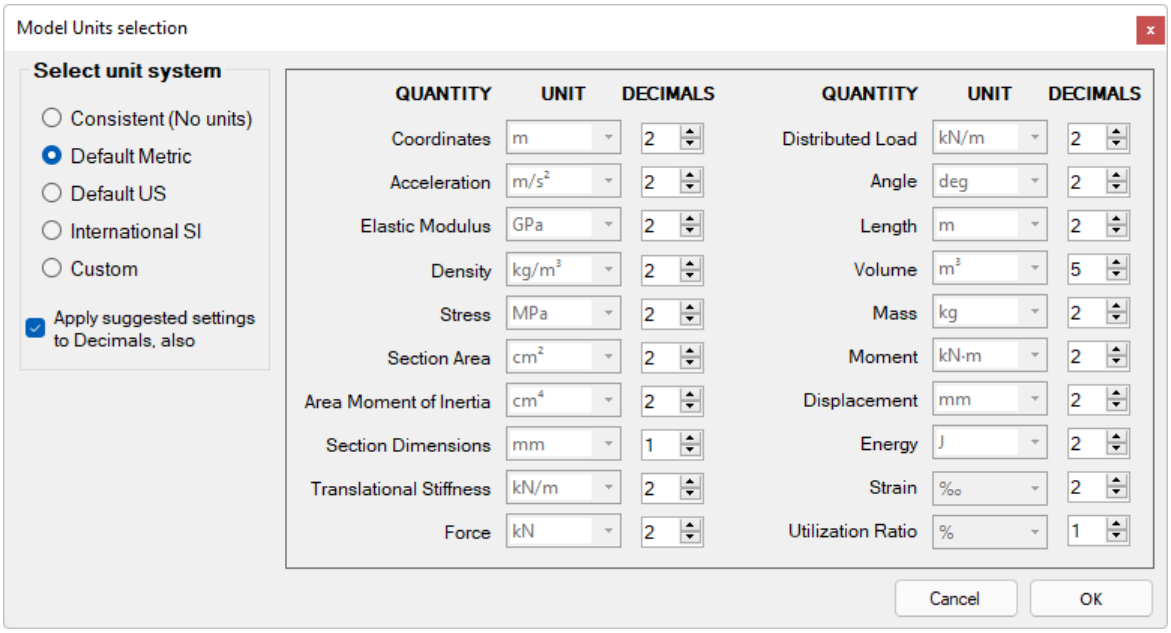
EngiLab Truss.2D supports the following predefined Systems of Units:

- **Default Metric**
- **Default US**
- **International SI**

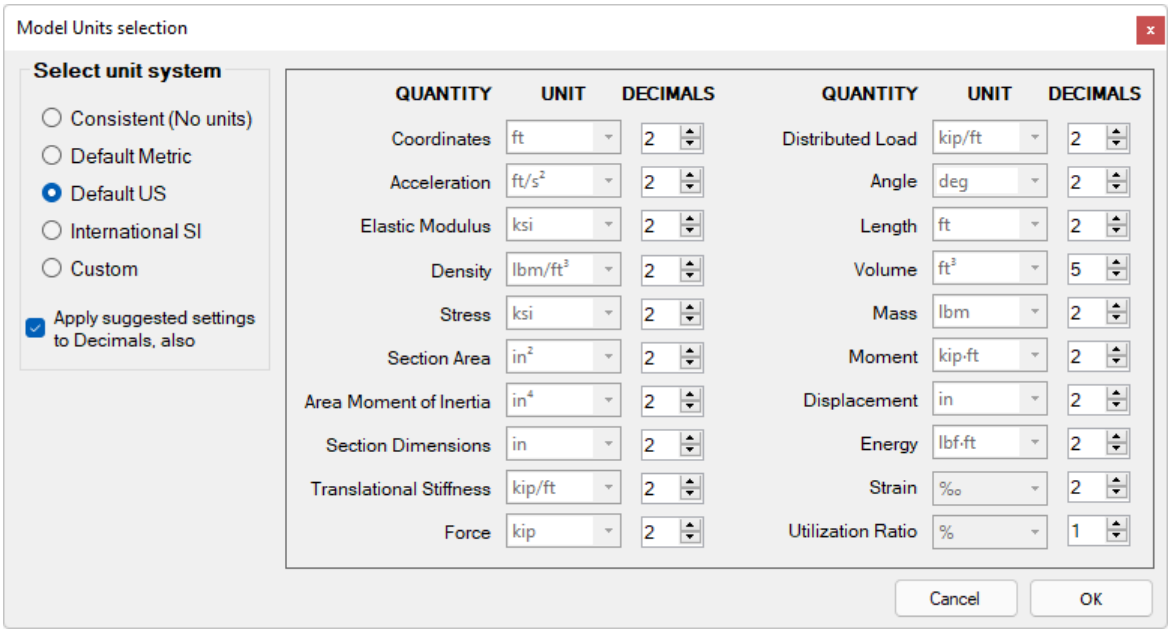
(c) 2022 EngiLab PC

The three systems are shown in the screenshots below. In these systems Units are "locked" and cannot be changed. The Decimal places for each quantity can be changed. These Decimal places are for illustration purposes, for showing the results or the Model data on screen and do not affect any calculations or the accuracy of the Analysis.

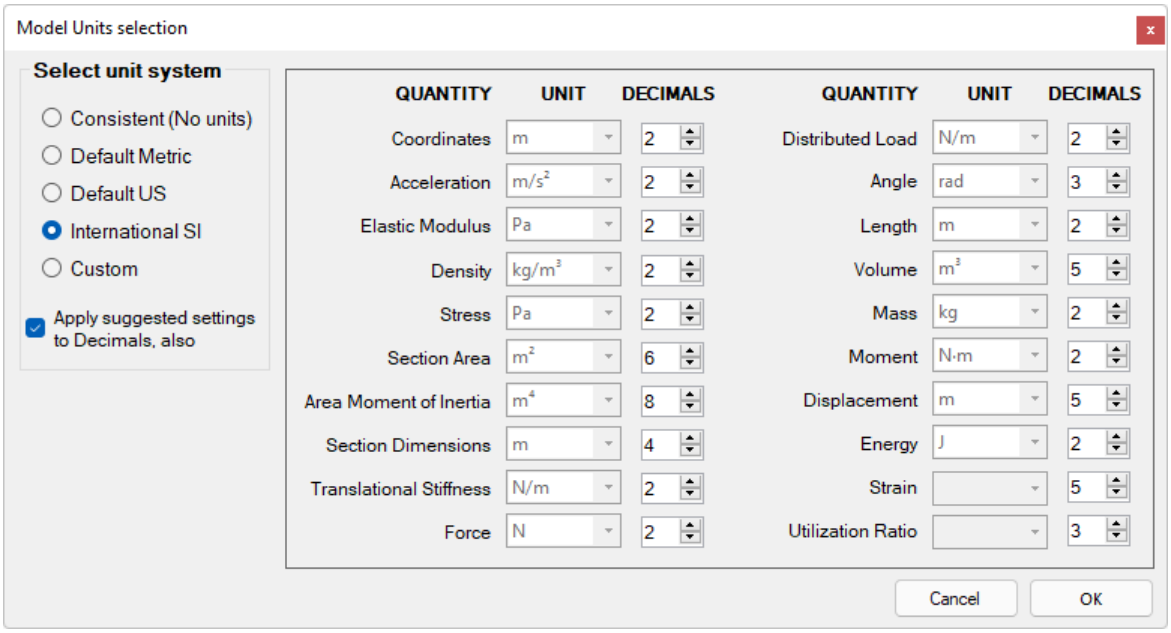
Tip: If you want to change a predefined System of Units, all you need to do is to select it and then click "**Custom**". This will unlock all fields and you can change any of them. The system will change to "Custom" but it will be based on the previously selected system.



Default Metric System of Units



Default US System of Units



International SI System of Units

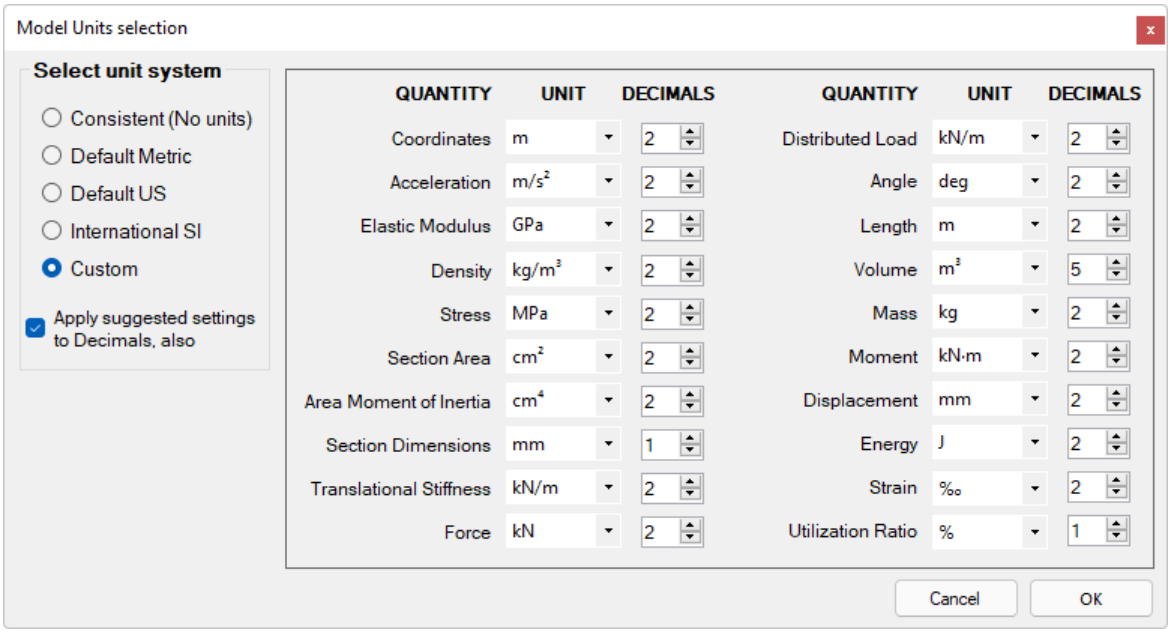
2.2.1.3 Custom System

The Custom System of Units is fully customizable by the user. A variety of units is offered for each unit category, for the user to select from.

Tip: You can first select a predefined system and then you can click "**Custom**". This will "unlock" all the unit fields and then you can choose whatever you want.

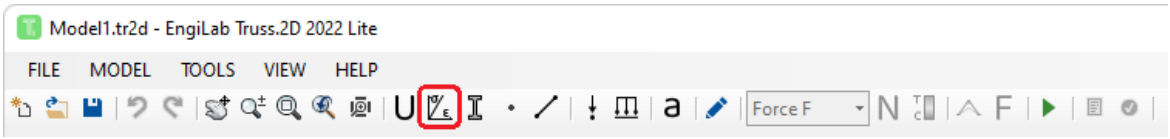
Note: Units cannot be left empty, with the exception of the last 2 (Strain and Utilization Ratio) which are unitless quantities. On the other hand, if ALL units are empty, this is acceptable as this is the definition of the Consistent System of Units. In other words, either ALL units can be empty (i.e. Consistent System) or no units can be empty (with the exception of the last 2).

(c) 2022 EngiLab PC



Custom System of Units

2.2.2 Materials



Each element is assigned a **Material**. The Material properties are the following:

- **Name** (optional, max. number of characters in Name: 20)
- **Elastic Modulus E** (or Modulus of Elasticity, or Young's Modulus) - Essential property
- **Density d** (optional, can be omitted, see below)
- **Yield Stress fy** (optional, can be omitted, see below)

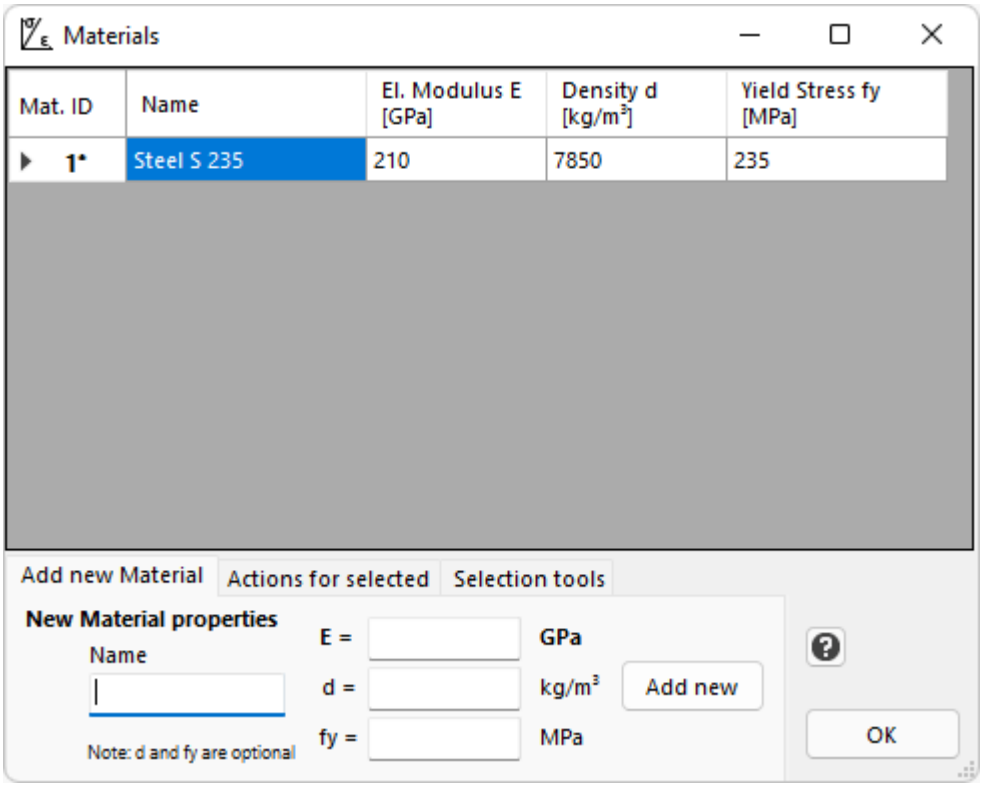
The **Elastic Modulus E** (essential property) of the Material is an important parameter affecting the Model Stiffness Matrix. It cannot be omitted and it cannot have a zero value.

The **Density d** (optional) is used in order for the program to calculate the Body Loads due to Linear Acceleration. For example, if you want to take into account the self-weight of Elements as an additional elemental load for each Element, then you have to provide the Material Density for the Material of each Element and also to define a Linear Acceleration Vector equal to the standard earth gravitational acceleration. If Density is omitted or it is equal to zero for a Material, then Elements that are assigned this Material will not take any Body (Acceleration) Loads even if a Linear Acceleration Vector is defined. By setting the Density equal to zero for a Material, you can model Weightless Elements.

(c) 2022 EngiLab PC

See also: Body (Acceleration) Loads.

The **Yield Stress fy** (optional) is used in order to check if the Element will fail due to yielding, after the analysis has been completed. This property is not used at all in the analysis of the Model. It is used when then Elements are checked when the utilization ratio for stress is calculated as the ratio σ/fy where σ is the stress of the Element (obtained from the analysis) and fy is the Yield Stress of the Material of the Element. If σ/fy≥1 then the Element fails the stress check.



Add new Material

To add a new Material, type the Name (optional), Elastic Modulus E, Density d (optional) and Yield Stress fy (optional) of the Material and click the "Add new Material" button.

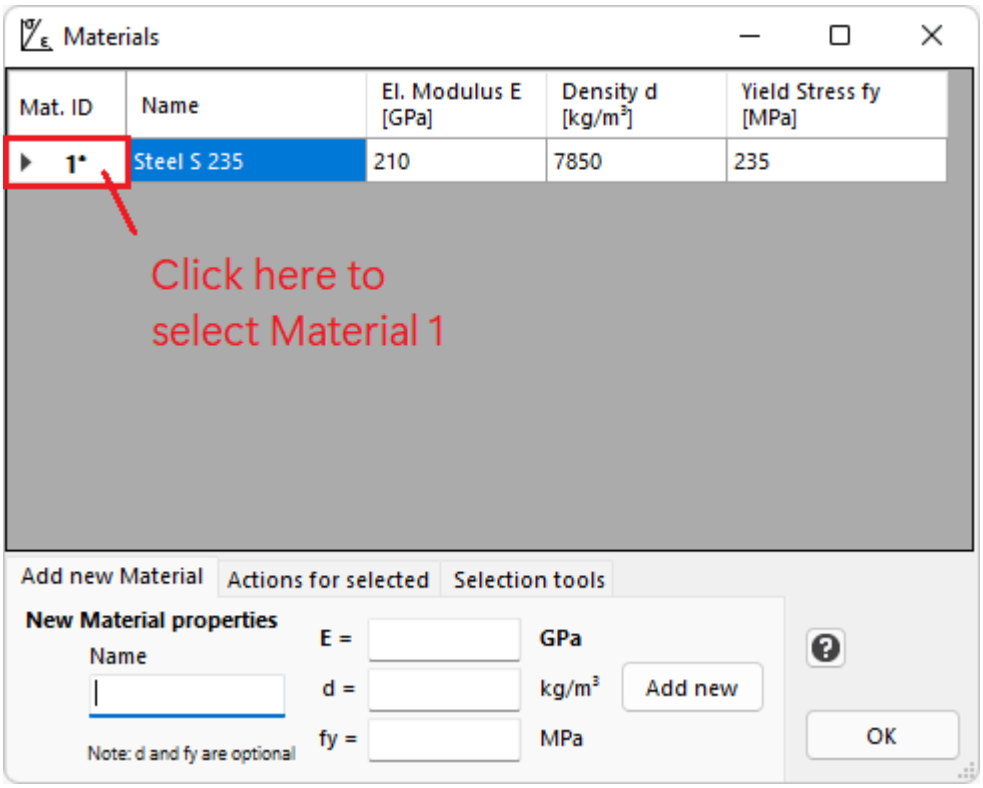
Edit an existing Material

You can click on the table and you can easily edit the properties of an already existing Material. Any change you make in the table is automatically reflected to the Model.

Select existing Material(s)

(c) 2022 EngiLab PC

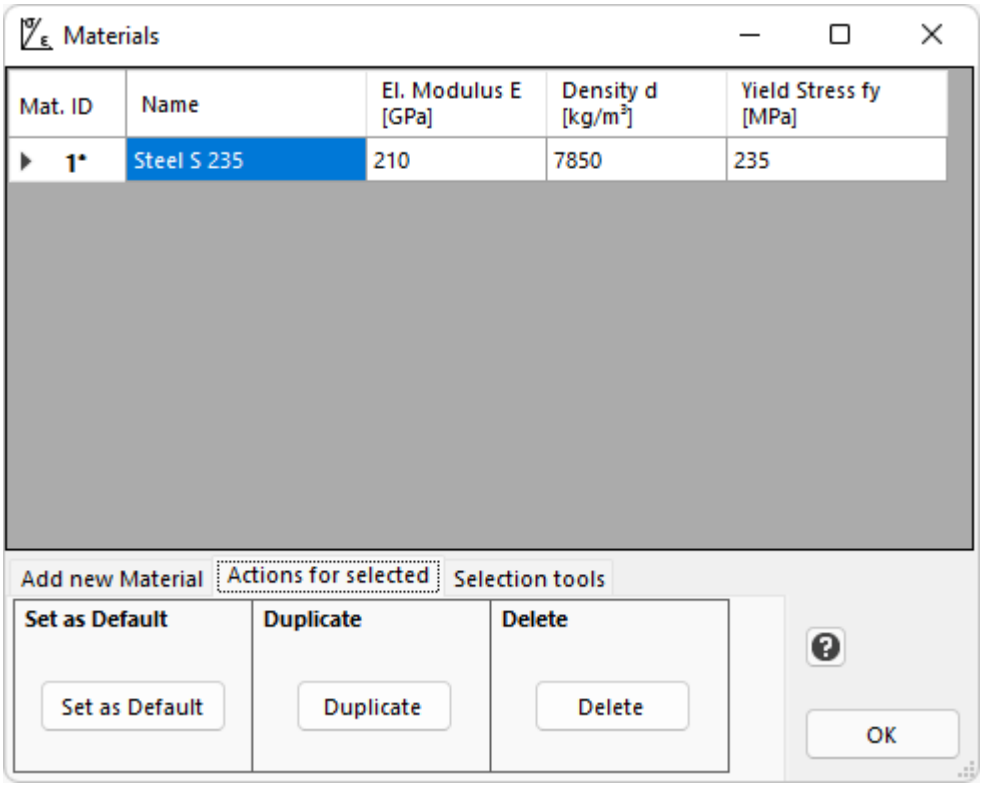
You can click on a row number to select one or more Materials. Then you can apply actions to the selected Materials, as shown next.



Actions for selected Materials

In order to perform these actions, **you have to first select the entire row(s) of the corresponding Material(s)** and then apply the action.

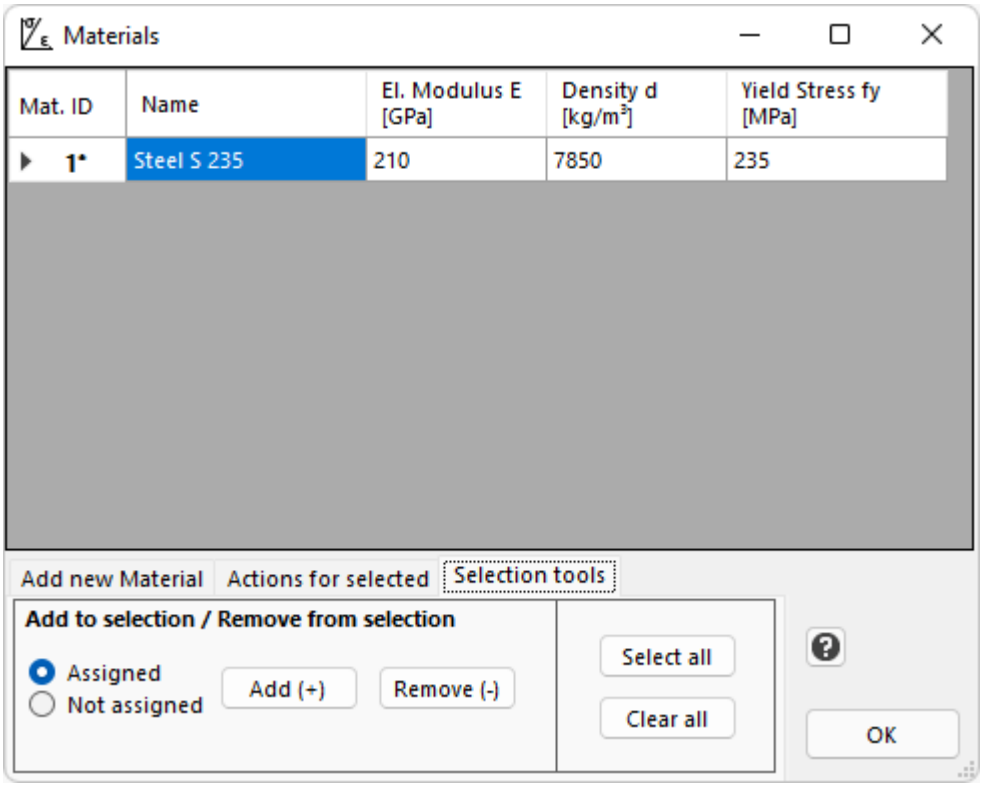
- **Set as Default**. Sets the selected Material as the Default Material. Any Elements drawn on the screen are assigned the Default Material.
- **Duplicate Material**(s). Click the "Duplicate" button. You can Duplicate more than one Materials at a time.
- **Delete Material**(s). Click the "Delete" button. You can Delete more than one Materials at a time.



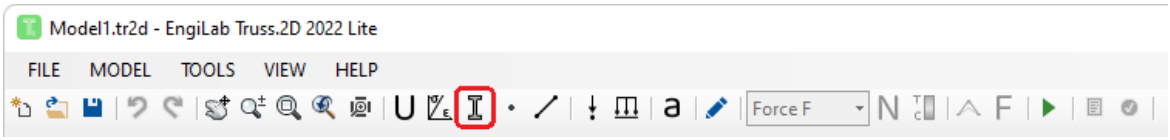
(c) 2022 EngiLab PC

Materials selection tools

This tab provides tools for the selection of Materials. For example, you can select (add to selection) or deselect (remove from selection) all the Materials that are assigned to Elements, or all the Materials that are NOT assigned to Elements.



2.2.3 Sections



Each element is assigned a **Section**. The Section properties are the following:

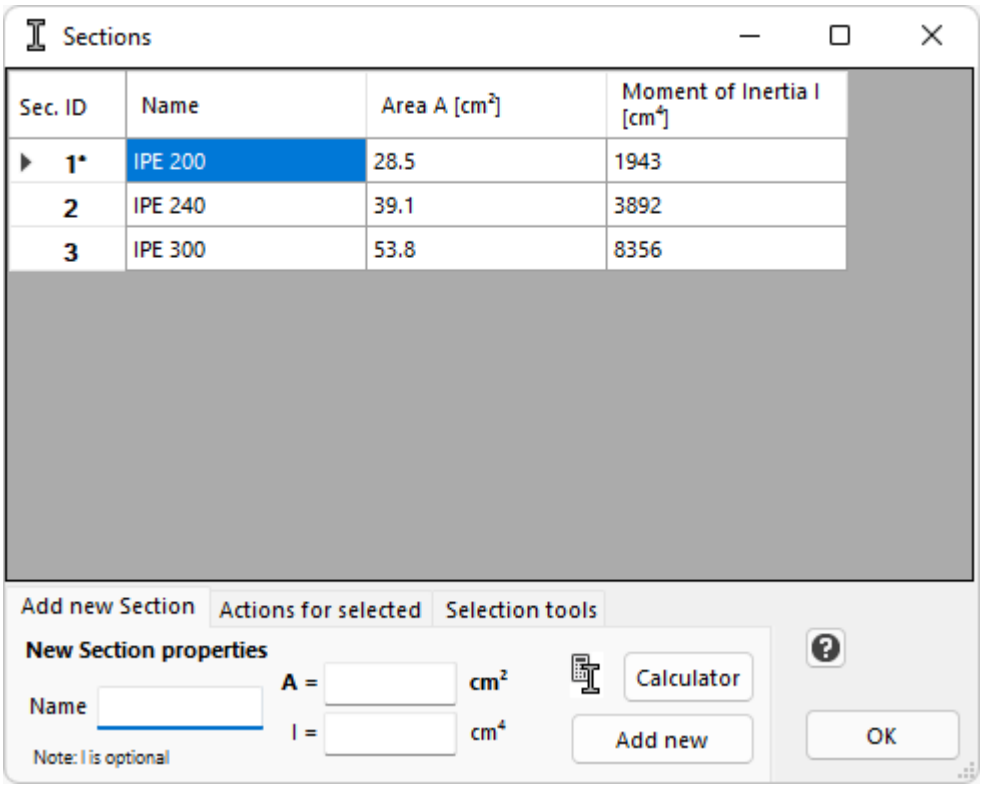
- **Name** (optional, max. number of characters in Name: 20)
- **Area A** - Essential property
- **Moment of Inertia I** (optional, can be omitted, see below).

The Moment of Inertia I of the Section is NOT used for the Analysis as it does not affect the stiffness matrix of a 2D Truss member.

This optional quantity is only used for checking the Element for Buckling. It is used for the calculation of the Euler's Critical Buckling Load, BL, based on the formula:

BL = π^2*E*I/L^2. For details, see the Element Checks section.

The **Area A** of the Section is an important parameter affecting the Model Stiffness Matrix.



Add new Section

To add a new Section, type the Name (optional), Area A and Moment of Inertia I (optional) of the Section and click the "Add new Section" button.

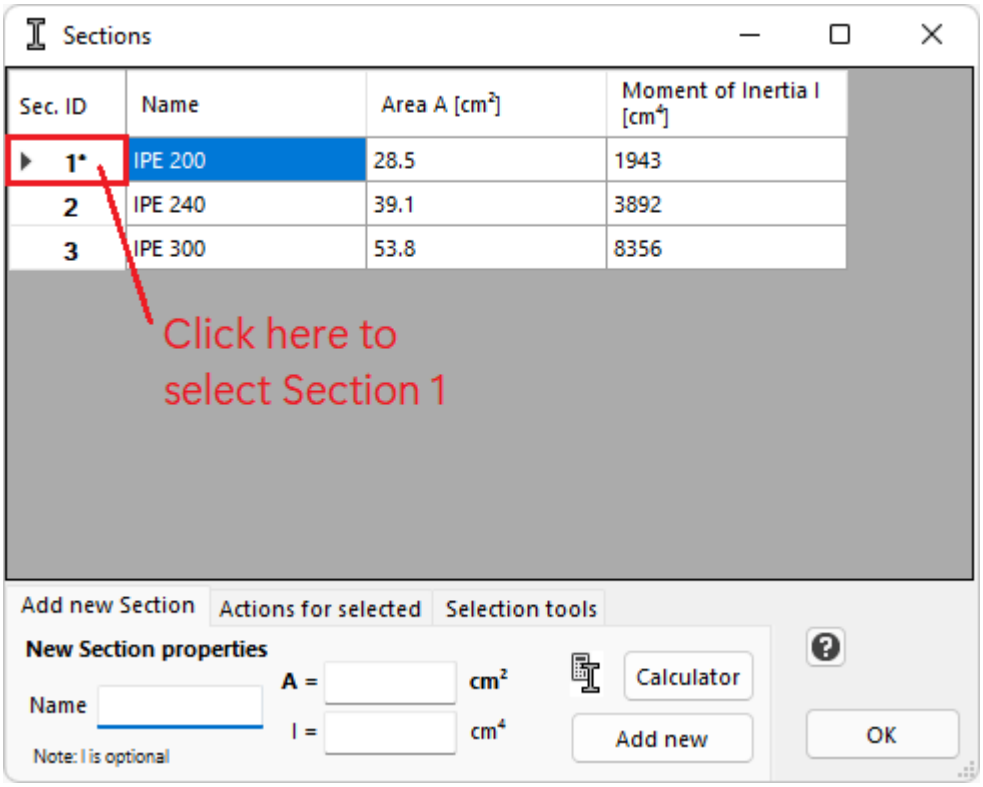
Edit existing Section(s)

You can click on the table and you can easily edit the properties of an already existing Section. Any change you make in the table is automatically reflected to the Model.

Select an existing Section

You can click on a row number to select one or more Sections. Then you can apply actions to the selected Sections, as shown in "Actions for selected Sections".

(c) 2022 EngiLab PC

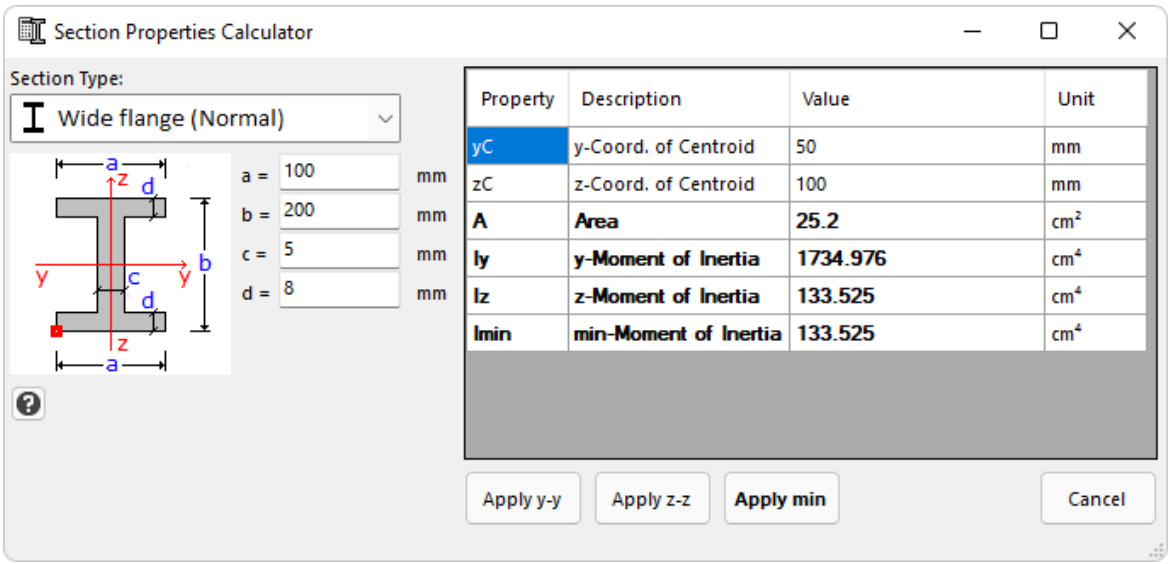

Sections
Sec. ID
Name
Area A [cm²]
Moment of Inertia I [cm⁴]
1*
IPE 200
28.5
1943
2
IPE 240
39.1
3892
3
IPE 300
53.8
8356
Click here to select Section 1
Add new Section
Actions for selected
Selection tools
New Section properties
Name
A =
cm²
I =
cm⁴
Note: I is optional
Calculator
Add new
OK

Section Properties Calculator

By clicking the "**Calculator**" button, a new form appears, as shown below. You can use the Section Properties Calculator to calculate the properties of various section shapes. The various section shapes are shown in the second figure below. The Calculator calculates various properties. The program needs only to take the **Area A** and **Moment of Inertia I** property.

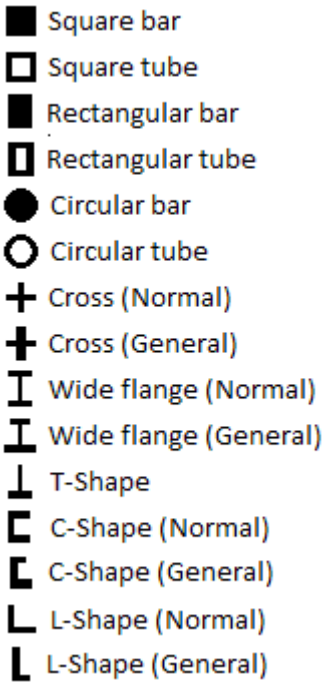
- Click "**Apply y-y**" to apply the Area A and the I_y Moment of Inertia
- Click "**Apply z-z**" to apply the Area A and the I_z Moment of Inertia
- Click "**Apply min**" to apply the Area A and the minimum Moment of Inertia, Imin

Note that the Moment of Inertia of the Section is not taken into account in the analysis of a truss structure. Only the Area A contributes to the axial stiffness of the member. The Moment of Inertia of the Section is only taken into account when the member is checked for buckling. If buckling can occur either in-plane of the 2D truss, or out-of-plane, then the minimum Moment of Inertia should be set.



The available sections are the following:

(c) 2022 EngiLab PC

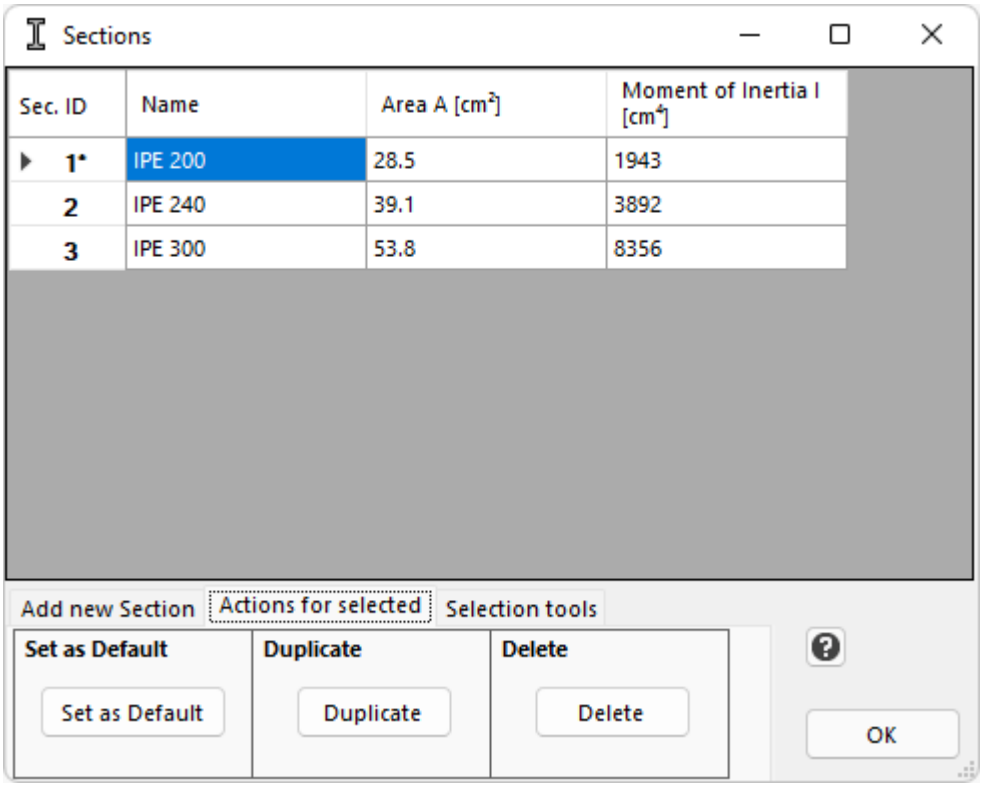


(c) 2022 EngiLab PC

Actions for selected Sections

In order to perform these actions, **you have to first select the entire row(s) of the corresponding Section(s)** and then apply the action.

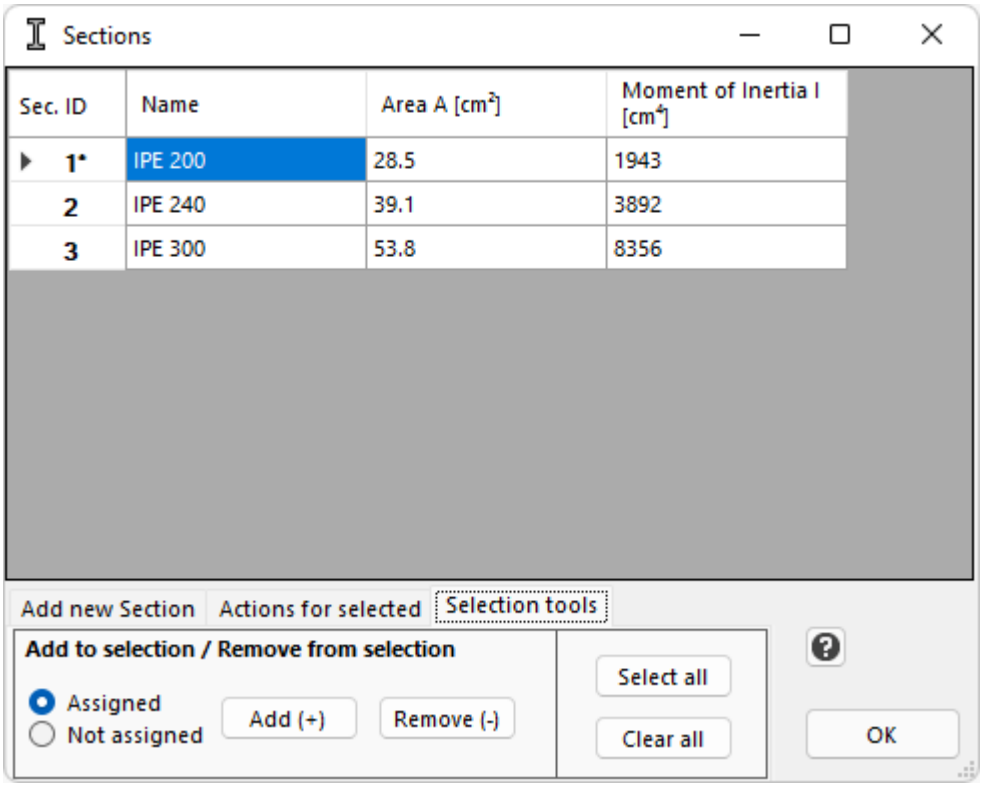
- **Set as Default**. Sets the selected Section as the Default Section. Any Elements drawn on the screen are assigned the Default Section.
- **Duplicate Section**(s). Click the "Duplicate" button. You can Duplicate more than one Sections at a time.
- **Delete Section**(s). Click the "Delete" button. You can Delete more than one Sections at a time.



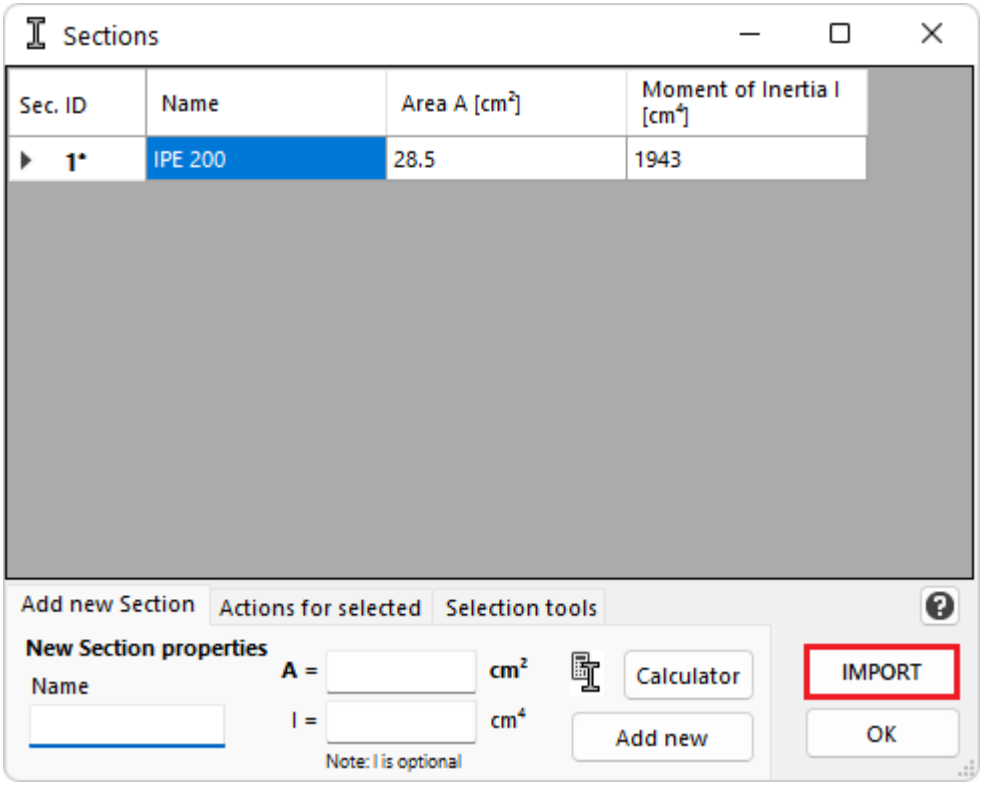
(c) 2022 EngiLab PC

Sections selection tools

This tab provides tools for the selection of Sections. For example, you can select (add to selection) or deselect (remove from selection) all the Sections that are assigned to Elements, or all the Sections that are NOT assigned to Elements.



2.2.3.1 Import Sections from Database

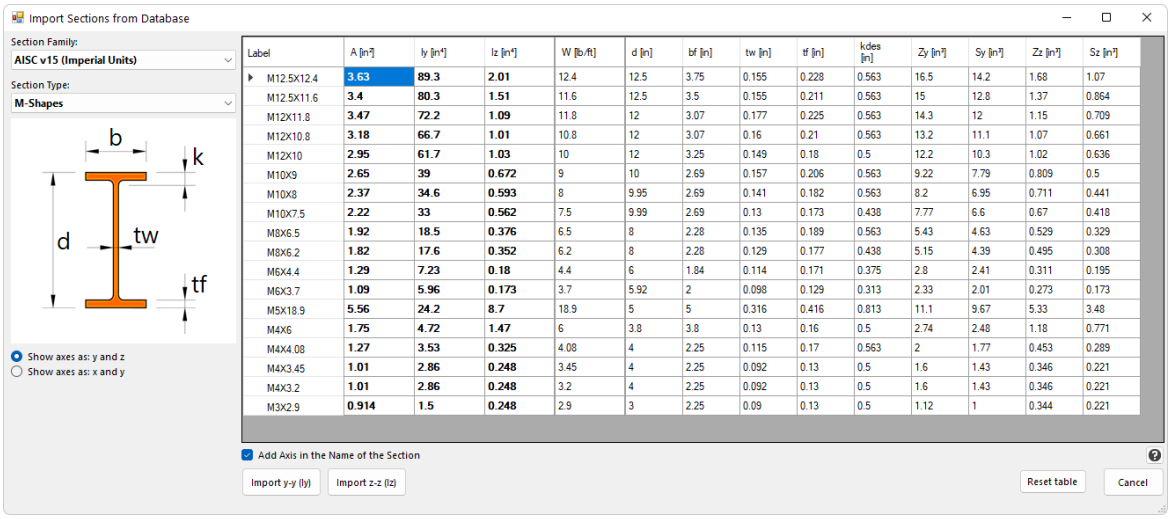


Click "**IMPORT**" to import sections from a Database. Currently, the following section Databases are supported by the program:

- **AISC v15 (Imperial Units)**, with 2091 sections (W-Shapes, M-Shapes, HP-Shapes, S-Shapes, C-Shapes, MC-Shapes, MT-Shapes, WT-Shapes, ST-Shapes, HSS_Rectangular, Angles_L, HSS_Round, PIPE, Double_Angles_2L)
- **AISC v15 (Metric Units)**, with 2091 sections (W-Shapes, M-Shapes, HP-Shapes, S-Shapes, C-Shapes, MC-Shapes, MT-Shapes, WT-Shapes, ST-Shapes, HSS_Rectangular, Angles_L, HSS_Round, PIPE, Double_Angles_2L)
- **European**, with 143 sections (IPE, IPE-A, IPN, HE-A, HE-B, HE-M, UPE)

By clicking **IMPORT** we get a new form.

(c) 2022 EngiLab PC

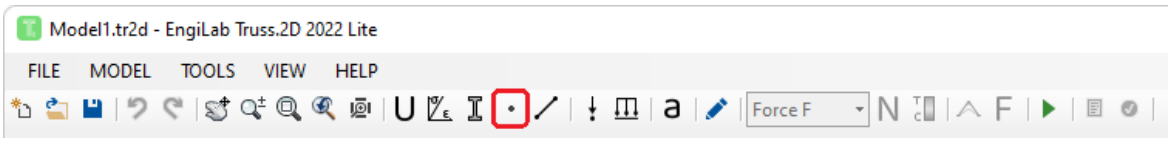


Label	A [in²]	Iy [in⁴]	Iz [in⁴]	W [lb/ft]	d [in]	bf [in]	tw [in]	tf [in]	kdes [in]	Zy [in³]	Sy [in³]	Zz [in³]	Sz [in³]
M12.5X12.4	3.63	89.3	2.01	12.4	12.5	3.75	0.155	0.228	0.563	16.5	14.2	1.68	1.07
M12.5X11.6	3.4	80.3	1.51	11.6	12.5	3.5	0.155	0.211	0.563	15	12.8	1.37	0.864
M12X11.8	3.47	72.2	1.09	11.8	12	3.07	0.177	0.225	0.563	14.3	12	1.15	0.709
M12X10.8	3.18	66.7	1.01	10.8	12	3.07	0.16	0.21	0.563	13.2	11.1	1.07	0.661
M12X10	2.95	61.7	1.03	10	12	3.25	0.149	0.18	0.5	12.2	10.3	1.02	0.636
M10X9	2.65	39	0.672	9	10	2.69	0.157	0.206	0.563	9.22	7.79	0.809	0.5
M10X8	2.37	34.6	0.593	8	9.95	2.69	0.141	0.182	0.563	8.2	6.95	0.711	0.441
M10X7.5	2.22	33	0.562	7.5	9.99	2.69	0.13	0.173	0.438	7.77	6.6	0.67	0.418
M8X6.5	1.92	18.5	0.376	6.5	8	2.28	0.135	0.189	0.563	5.43	4.63	0.529	0.329
M8X6.2	1.82	17.6	0.352	6.2	8	2.28	0.129	0.177	0.438	5.15	4.39	0.495	0.308
M6X4.4	1.29	7.23	0.18	4.4	6	1.84	0.114	0.171	0.375	2.8	2.41	0.311	0.195
M6X3.7	1.09	5.96	0.173	3.7	5.92	2	0.098	0.129	0.313	2.33	2.01	0.273	0.173
M5X18.9	5.56	24.2	8.7	18.9	5	5	0.316	0.416	0.813	11.1	9.67	5.33	3.48
M4X6	1.75	4.72	1.47	6	3.8	3.8	0.13	0.16	0.5	2.74	2.48	1.18	0.771
M4X4.08	1.27	3.53	0.325	4.08	4	2.25	0.115	0.17	0.563	2	1.77	0.453	0.289
M4X3.45	1.01	2.86	0.248	3.45	4	2.25	0.092	0.13	0.5	1.6	1.43	0.346	0.221
M4X3.2	1.01	2.86	0.248	3.2	4	2.25	0.092	0.13	0.5	1.6	1.43	0.346	0.221
M3X2.9	0.914	1.5	0.248	2.9	3	2.25	0.09	0.13	0.5	1.12	1	0.344	0.221

Select entire rows from the table and then click "**Import y-y (Iy)**" or "**Import z-z (Iz)**" to import the section to the Model.

- "**Import y-y (Iy)**": Imports the Section Name, Area A and **Moment of Inertia Iy** (usually the strong axis of the section)
- "**Import z-z (Iz)**": Imports the Section Name, Area A and **Moment of Inertia Iz** (usually the weak axis of the section)

2.2.4 Nodes



Nodes connect Elements. Each Element has a Start Node (i) and an End Node (j). Each node is located at a (**X, Y**) Point (Global Axes) in the 2D plane and it has 2 Degrees Of Freedom (DOFs) for a 2D Truss:

- X-Displacement
- Y-Displacement

In 2D Trusses, the rotational DOF is not taken into account (unlike 2D Frames where it is important to take it into account) and all members can freely rotate at their ends.

(c) 2022 EngiLab PC

Each DOF can be:

- Free to move (No support, No spring)
- Fixed (Fully constrained, not able to move at all)
- With a spring on (Given the spring stiffness, the spring provides a reaction force that is proportional to the displacement of the corresponding DOF).

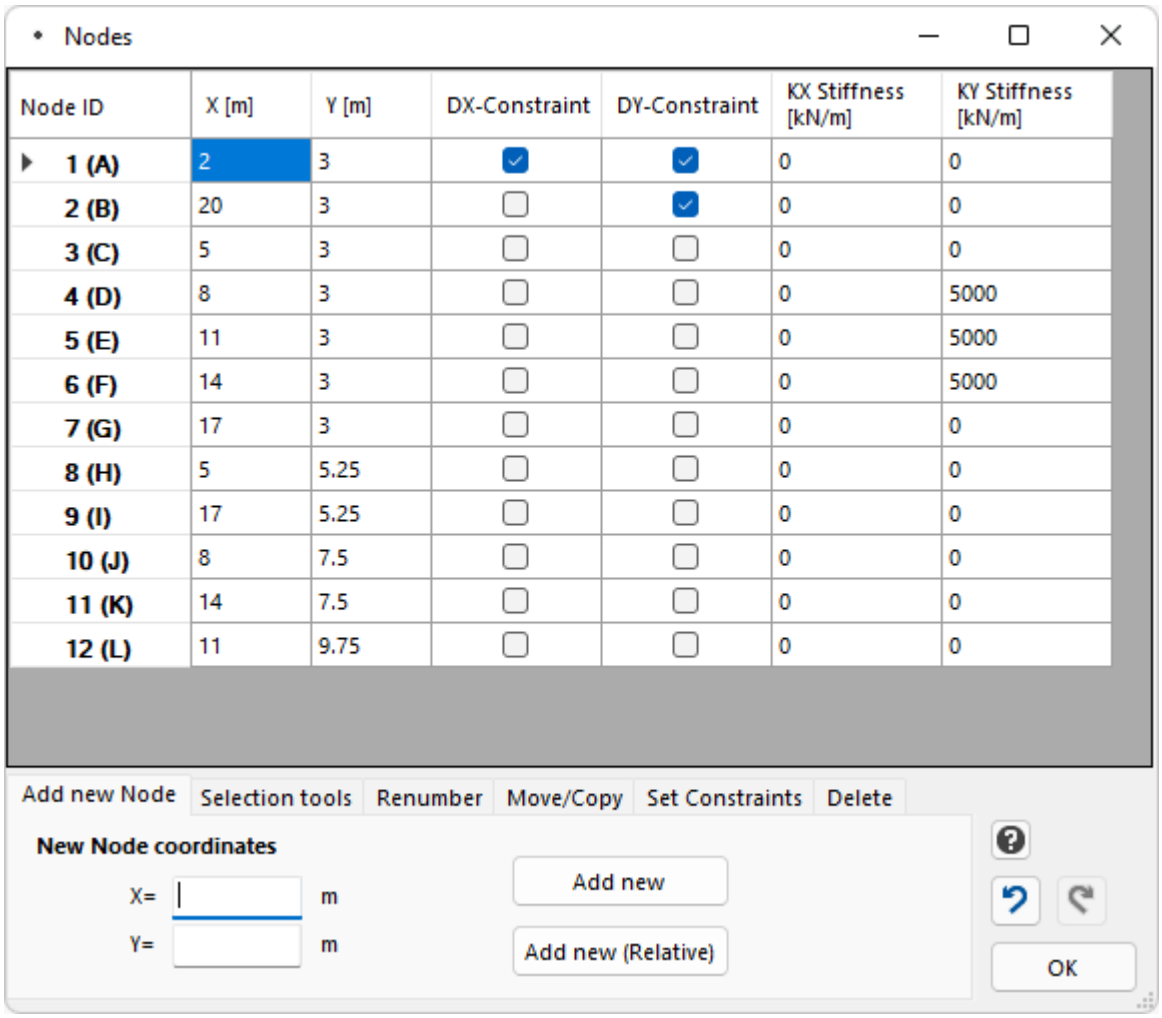
Note that a DOF with a spring on is at an intermediate state between Free and Fixed. If the spring stiffness is zero, then it will behave exactly as a free DOF, while if the spring stiffness reaches infinity, it will behave as a fixed DOF. Any value in between zero and infinity will provide some stiffness.

(c) 2022 EngiLab PC

Node properties

The Node properties are the following:

- **Coordinates: X, Y**
- **Constraints: DX-Constraint, DY-Constraint**. Each one can be Checked or Not checked.
- **Springs stiffness: KX Stiffness, KY Stiffness**. Each stiffness component has a real positive value, or a zero value. The stiffness cannot have a negative value.



Coordinates

Each node is located at a (**X, Y**) Point (Global Axes) in the 2D plane.

Constraints

The properties **DX-Constraint**, **DY-Constraint** determine whether the corresponding DOF (DX or DY) of the Node is constrained or not. DX and DY are the X and Y displacements of the node, respectively.

(c) 2022 EngiLab PC

Each DOF can be:

- Free (0 = Unchecked), or
- Fully constrained (1 = Checked).

(c) 2022 EngiLab PC

As a result, there are in total 4 types of Nodes in a 2D Truss, as follows:

Picture	DX-Constraint	DY-Constraint	Description
	NO	NO	Free Node (00)
	✓	NO	y-Roller (10)
	NO	✓	x-Roller (01)
	✓	✓	Pinned (11)

Springs

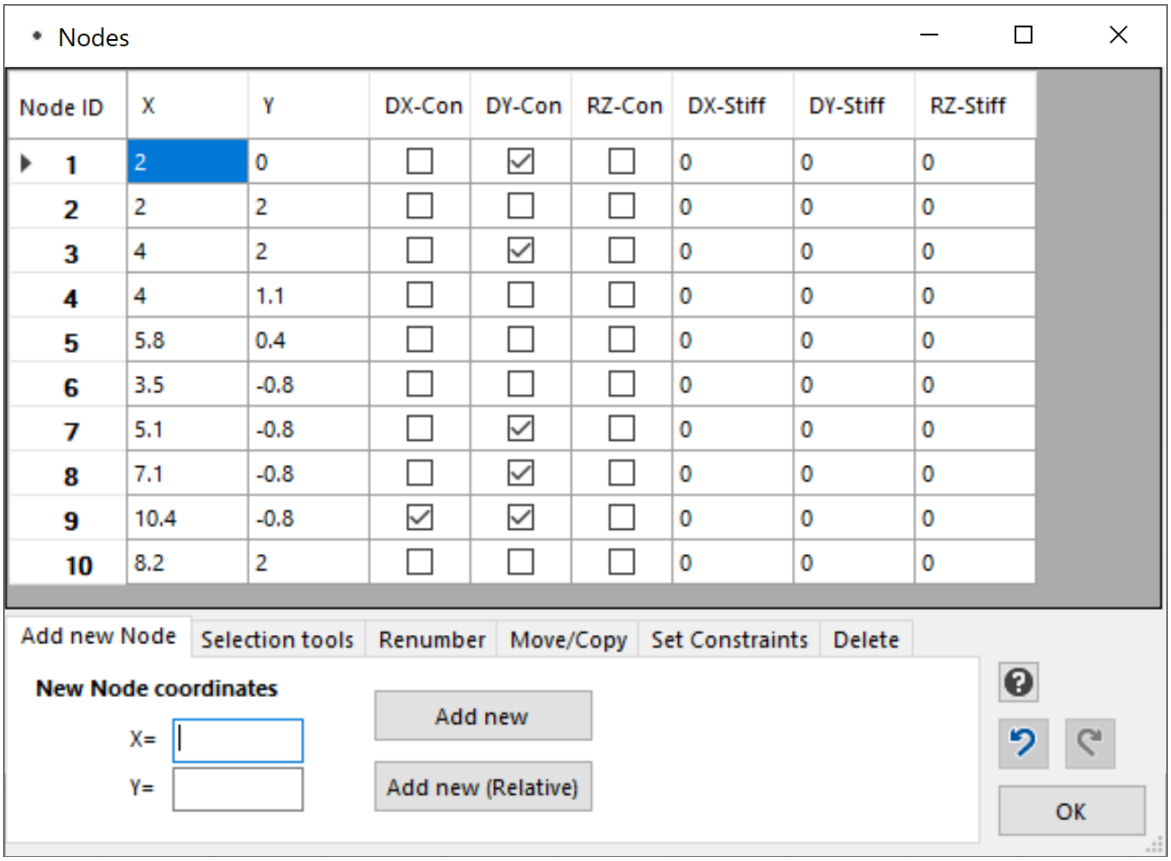
The properties **KX Stiffness, KY Stiffness** determine the Stiffness of the Spring (Elastic constant) of the corresponding DOF. A spring provides a spring reaction force that is proportional to the displacement of the corresponding DOF. Each stiffness component has a real positive value, or a zero value. The stiffness cannot have a negative value.

Note: A DOF that has a spring should not be constrained, as there is no point in that - Any constrained DOF will not have a displacement and as a result the spring reaction will be zero. In any case, if for a given DOF there is a spring and it is also constrained, then the spring is ignored and the DOF is handled as fully constrained.

Add new Node

To add a new Node, you can type the X and Y Coordinates of the Node and click the "Add new" button. This uses the **absolute coordinates X, Y** for the definition of the new Node.

(c) 2022 EngiLab PC



Another convenient way to define a new Node is by using relative coordinates, based on the coordinates of a previously selected Node. For example, you select Node (row) 10, you set the X and Y values, and then you click "**Add new (Relative)**". The new Node is defined in relation to the previously selected Node. This is very convenient when defining Nodes with a fixed distance between them. In this case, the previous Node acts as the selected Node, so you can keep adding Nodes using relative coordinates based on the previously defined Node.

Note: If multiple Nodes are selected and the "Add new (Relative)" button is clicked, then the Node with the highest ID (of the selected ones) is taken into account as the reference Node for the definition of the new Node.

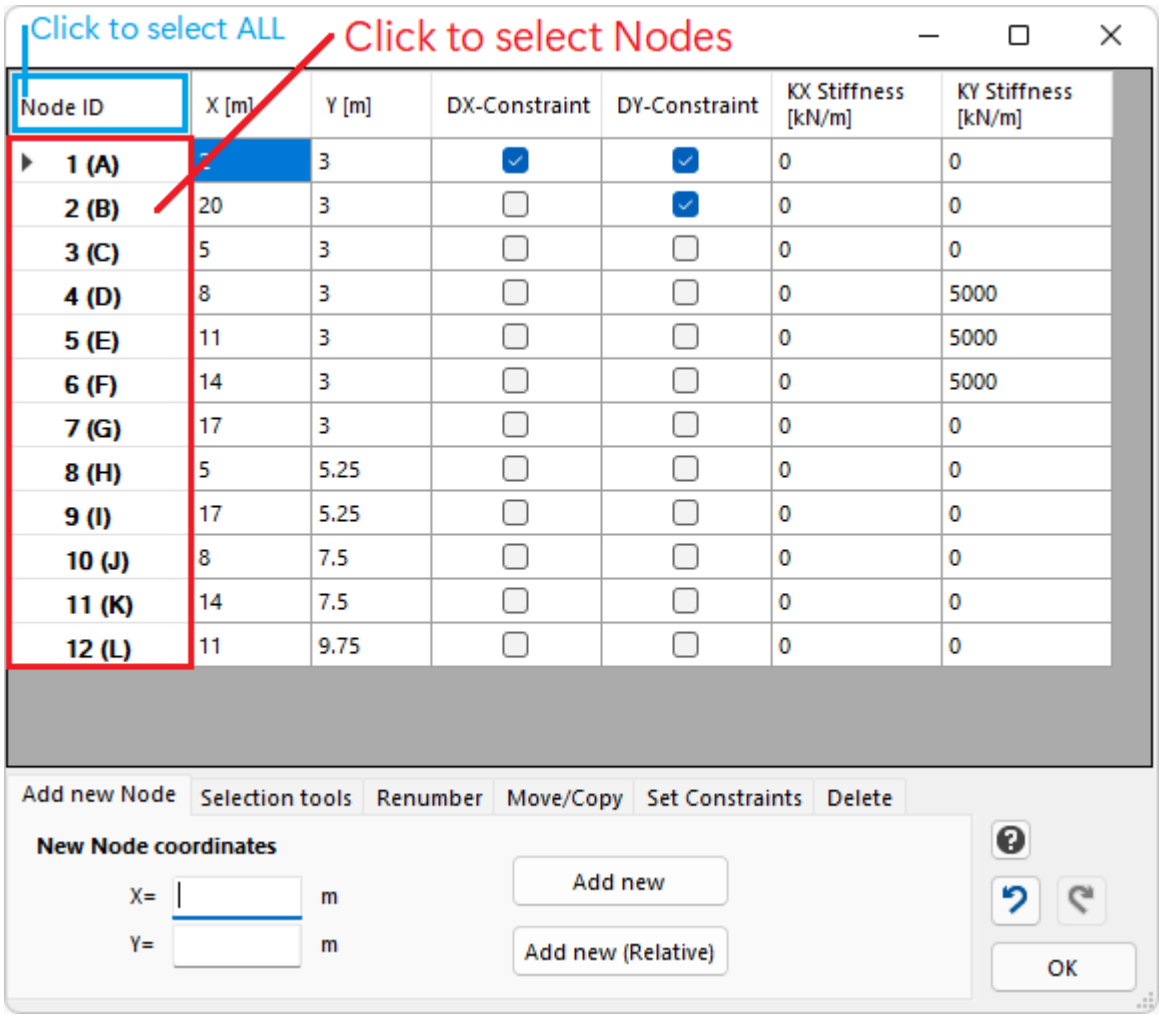
Edit an existing Node

You can click on the table and you can easily edit the properties of an already existing Node. Any change you make in the table is automatically reflected to the Model on the screen.

Select existing Node(s)

You can click on a row number to select one or more Nodes. Then you can apply actions to the selected Nodes (renumber, move/copy, set constraints, delete, etc).

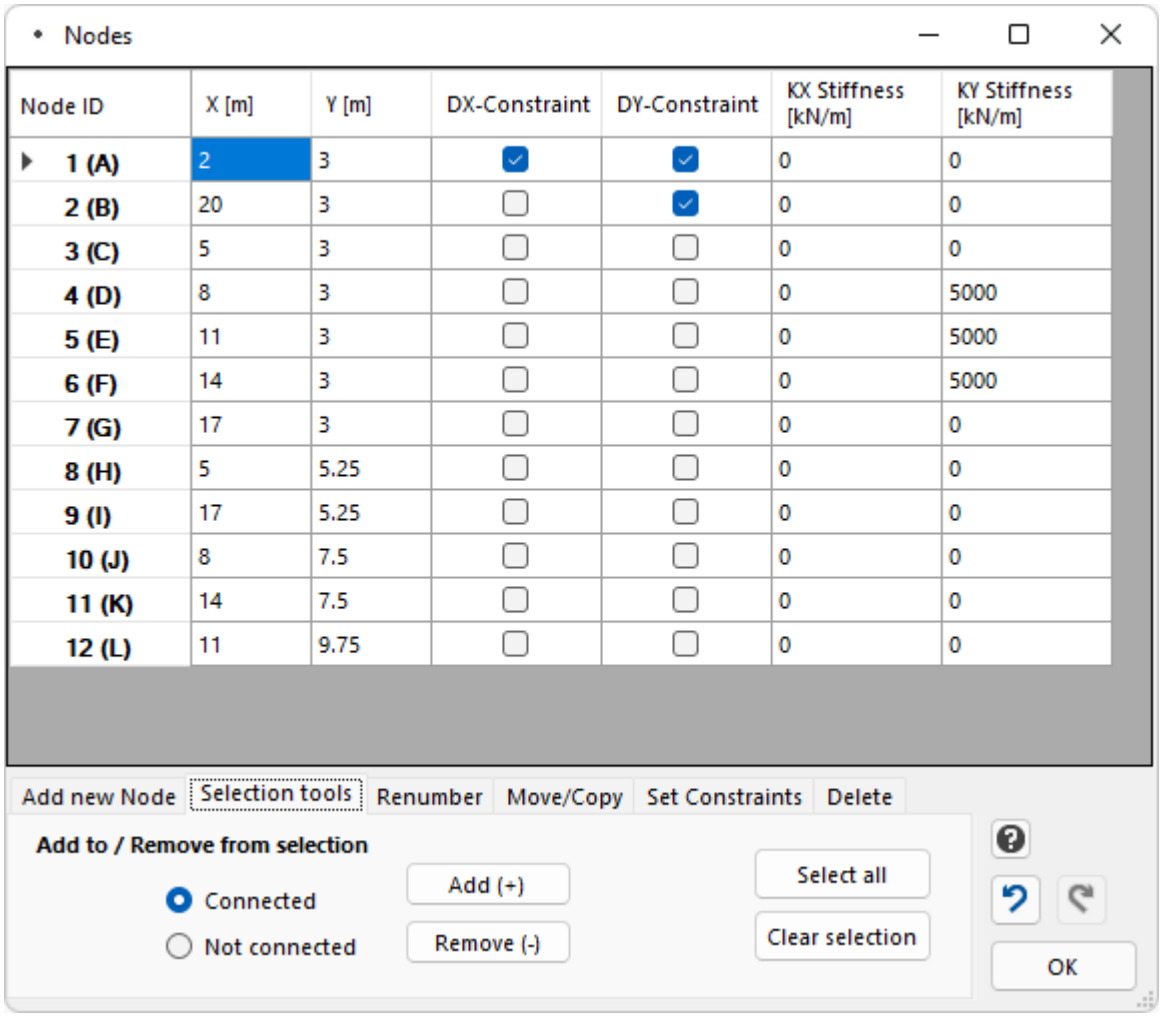
(c) 2022 EngiLab PC



Selection tools

This tab provides tools for the selection of Nodes. For example, you can select (add to selection) or deselect (remove from selection) all the Nodes that are connected to Elements, or all the Nodes that are NOT connected to Elements. This is useful when for example you need to Delete all the Nodes of the Model that are not connected to Elements. You can also Select all Nodes, or Clear the selection.

(c) 2022 EngiLab PC



Renumber Nodes

In order to perform this action, **first you have to select the entire row(s) of the corresponding Node(s)** and then apply the action.

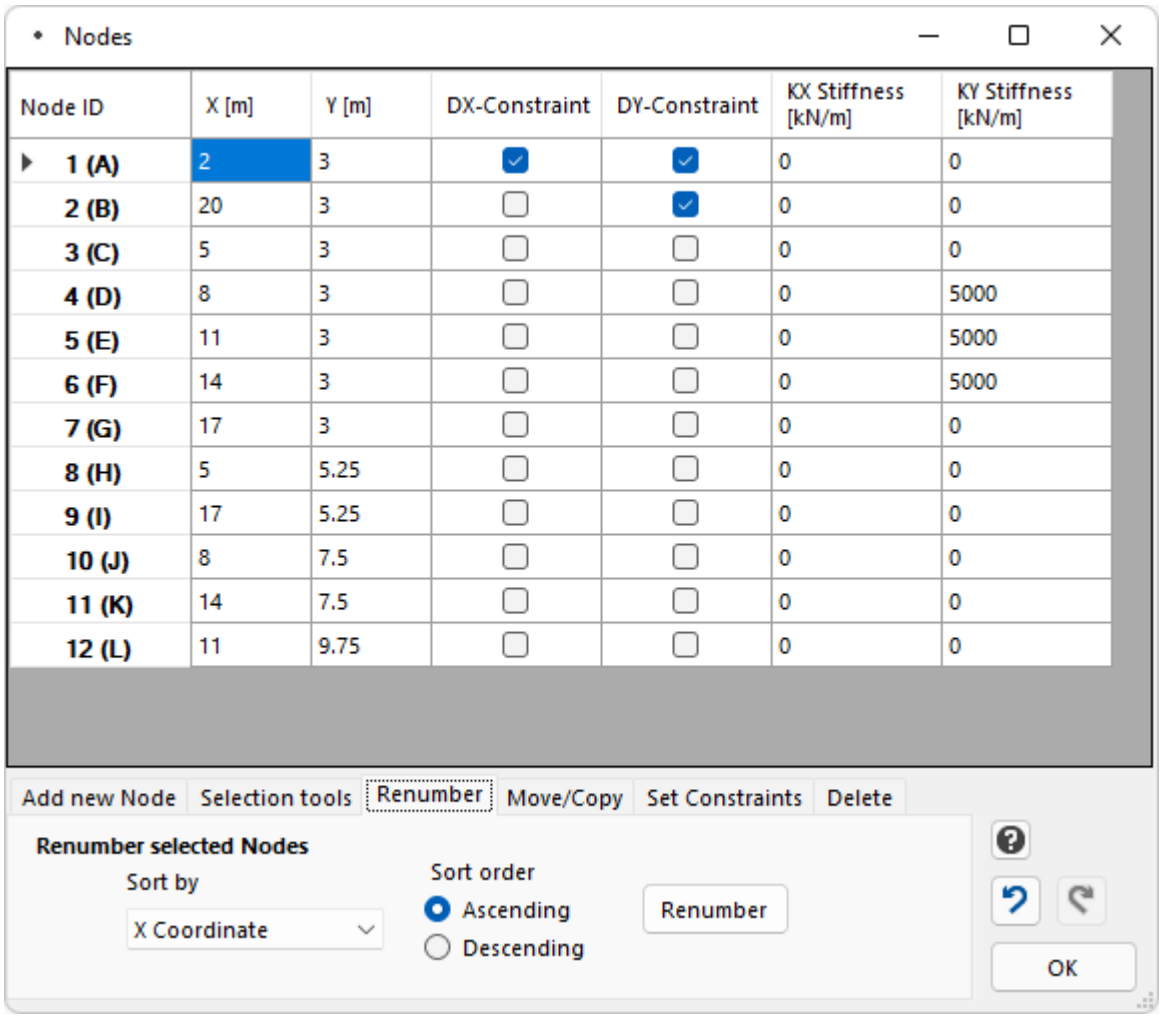
A tool is available for the renumbering of (the selected) Nodes, based on various criteria:

- X Coordinate
- Y Coordinate
- (X + Y)
- (X - Y)
- (X^2 + Y^2)

First, the Nodes have to be selected. Then we select the "Sort by" field and then the sort order (ascending or descending). Finally, we click the "Renumber" button.

(c) 2022 EngiLab PC

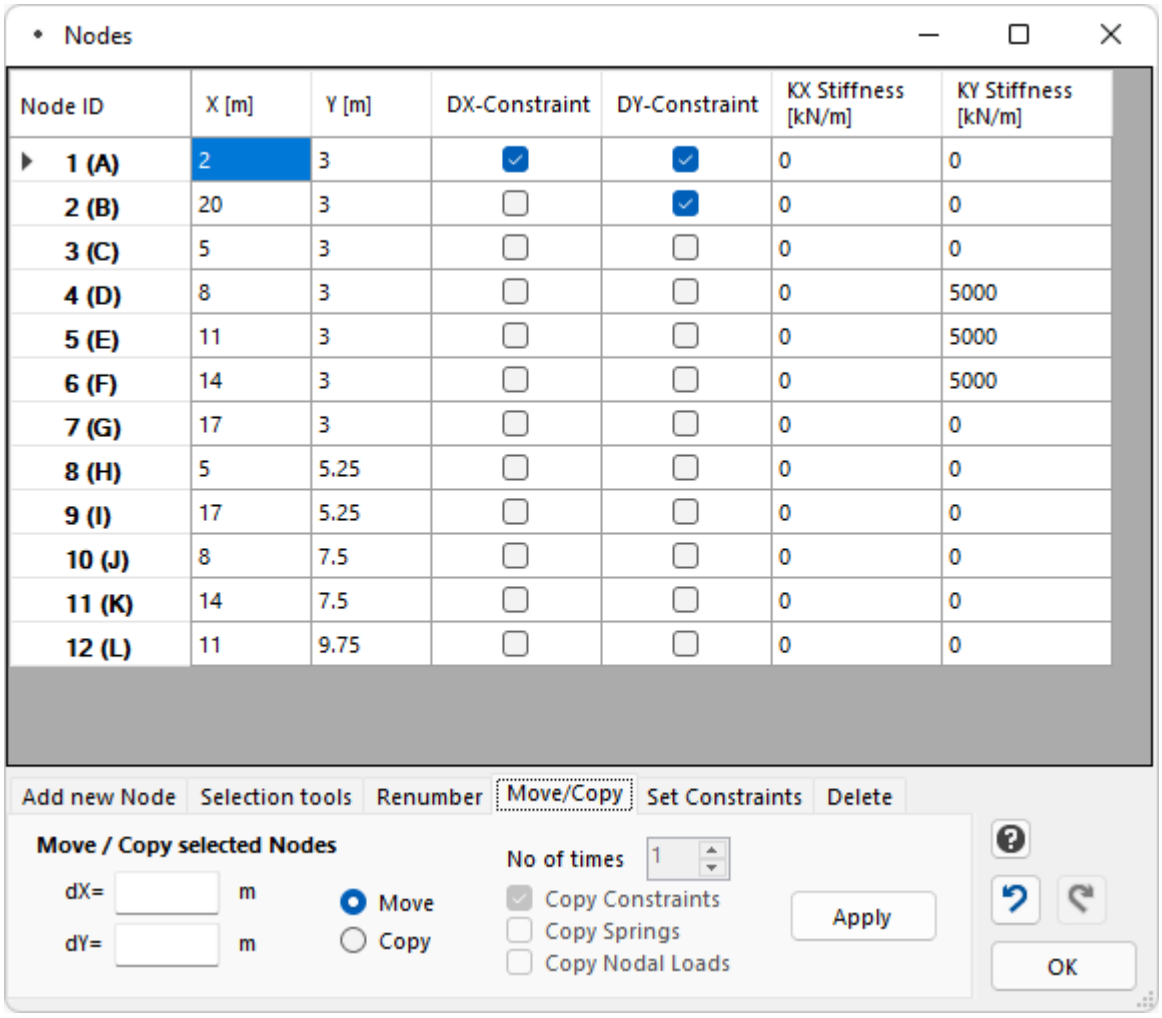
Important: Renumbering is done for the selected Nodes. The non-selected Nodes remain unchanged. To renumber all Nodes, all Nodes have to be selected first.



Move/Copy Nodes

In order to perform these actions, **first you have to select the entire row(s) of the corresponding Node(s)** and then apply the action.

This tool is used for moving or copying Nodes. The user specifies the Move/Copy Vector {V}={dX, dY}.



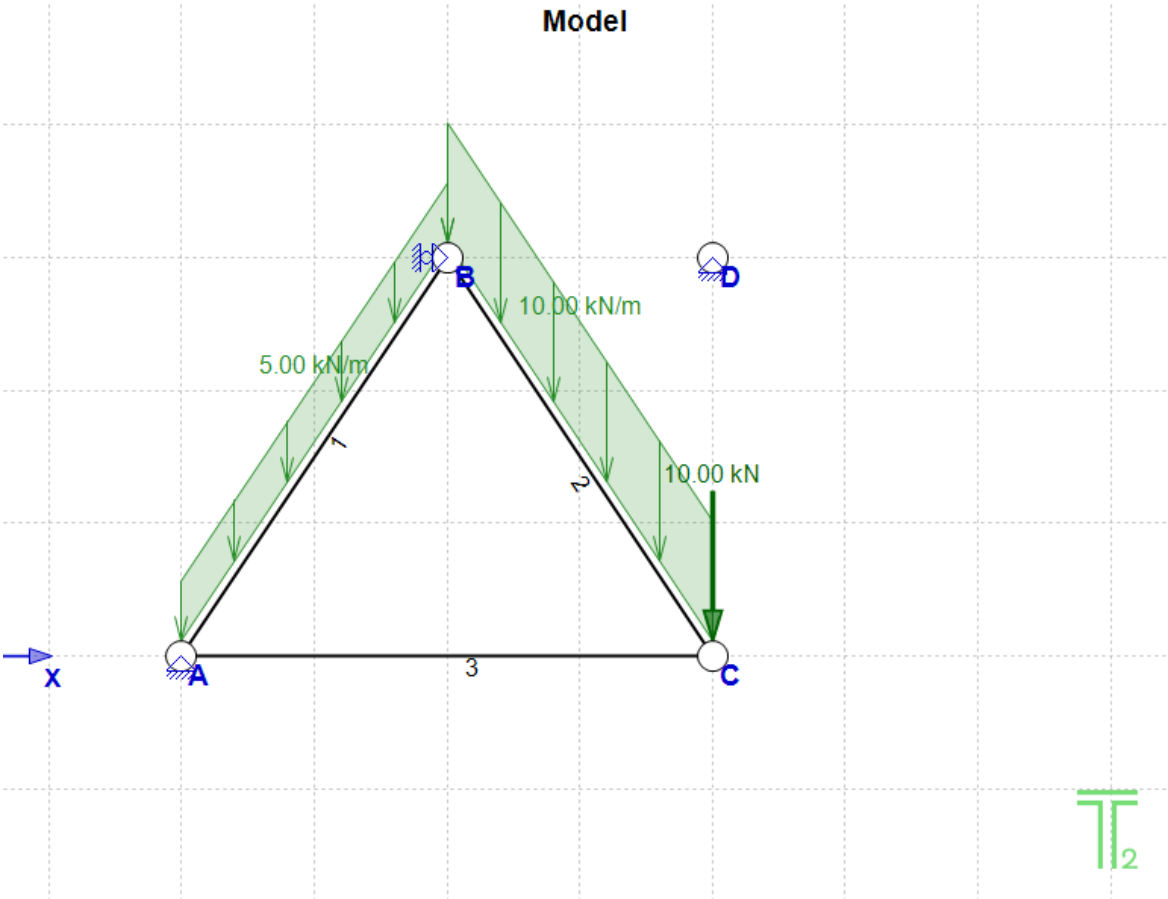
Move Nodes

- Multiple Nodes (one or more) can be selected to be moved at once.
- Move Nodes can only be applied once (only one time), not a number of times (only copy works with the option to apply it over a number of times).
- With the Move command, several Nodes can be moved to a new location. The user defines the move vector as {dX, dY}.
- Nodes are moved together with their supports, springs and any nodal loads on them.
- If another Node is already at the new location, it is not affected. In the end there will be two Nodes with different IDs, at the same location.
- If elements are connected to the Nodes that move, they will be moved also together with the Nodes, as Elements depend on their Nodes for their start and end points.
- If both Nodes (start Node i and end Node j) of an Element move, then the Element itself is moved without any change in its length or its other properties. The Element simply has a new position in the 2D plane, as both its Nodes move the same.

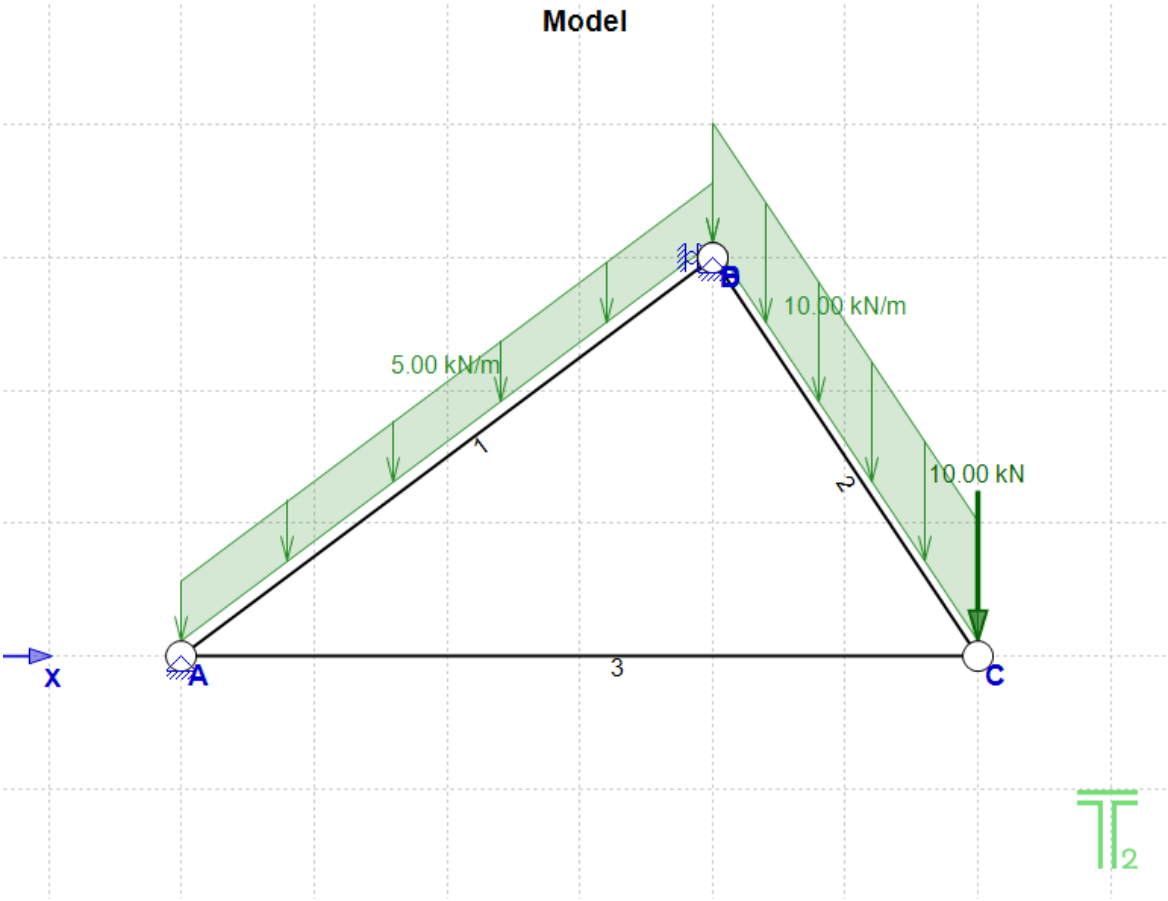
(c) 2022 EngiLab PC

- If only one Node (either the start Node i or the end Node j) of an Element moves, then the Element's length will change. An Elemental load on the Element will not change its value (as it is defined per unit length of the element), but the total load (total force) on the element will do change, due to the change in the length of the Element.

Example 1 – Moving Nodes



In the above example, Nodes **B** and **C** are moved with the vector {**DX=2, DY=0**}. The result of the operation is the following:



- Note the change in the length of Element 1 because of Node B moving to point (6, 3) from point (4, 3).
- The length of Element 2 is not affected as both its Nodes (start and end) have moved the same.
- The uniform load on Element 1 remains 5 kN/m (as force per unit length), but the total load on Element 1 has now changed to 5*5 = 25 kN from 5*3.61 = 18.05 kN as it now refers to the new length of the element (5.00 m), which is now increased (from 3.61 m before).
- Node B has moved to (6, 3) where Node D was already previously. Node D is not affected and in the end, there will be two Nodes (B and D) at the same location (6, 3).

Copy Nodes

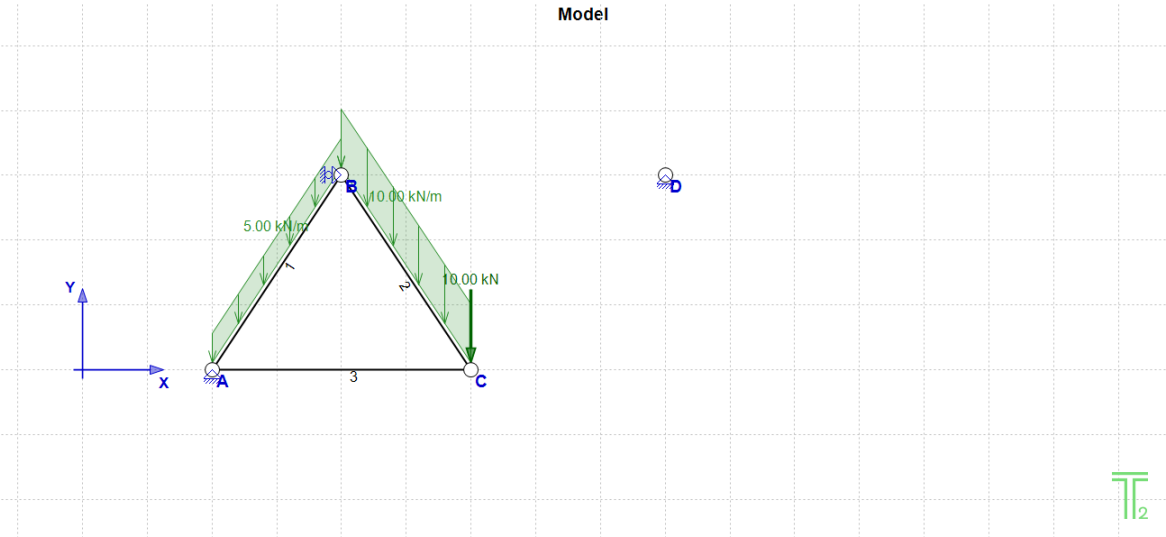
- Multiple Nodes (one or more) can be selected to be copied at once.
- Copy Nodes can be applied multiple times (up to 50) with a single click.
- With the Copy command, several Nodes can be copied to a new location. The user defines the copy vector as {dX, dY} and the number of times the operation will be applied.

(c) 2022 EngiLab PC

- If the corresponding check boxes are checked, Nodes are copied together with their Constraints (supports), Springs and Nodal Loads (forces or moments)
- By default, the "Copy Constraints" and the "Copy Springs" check boxes are checked, while the "Copy Nodal Loads" check box is NOT checked. All these can be changed by the user.
- If another Node is already at the new position, it is not affected. In the end there will be two Nodes with different IDs, at the same position.
- Copy Nodes does not affect any Elements, as elements depend on existing Nodes only.

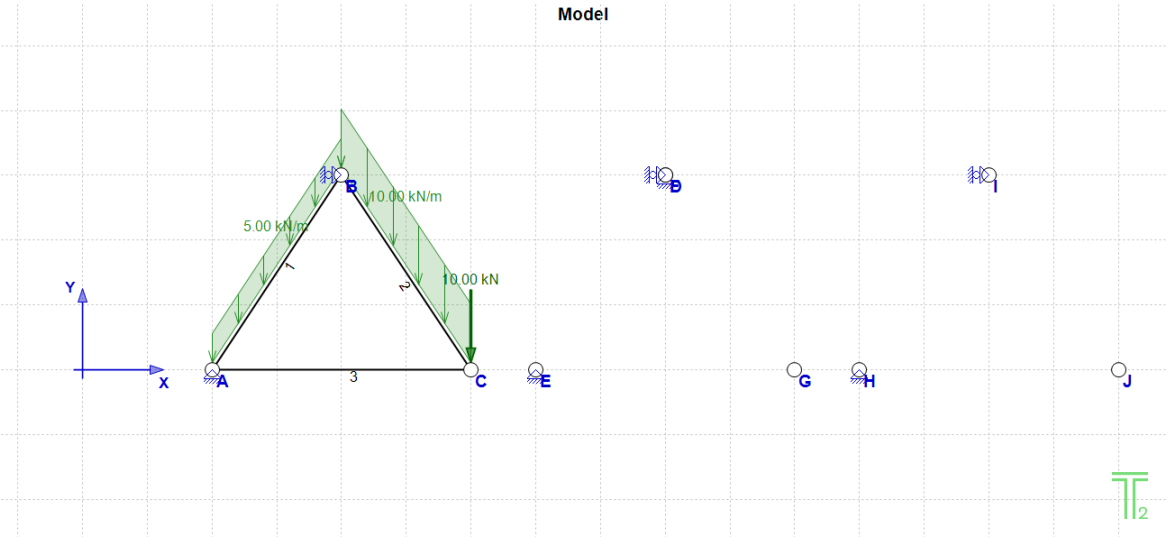
Example 2 – Copying Nodes

Copy Nodes works similarly with Move Nodes, but this time new Nodes are created instead of moving already existing ones. In the previous model of Example 1:



We now copy Nodes A, B and C with the vector {DX=5, DY=0}, 2 times, with the following copy options: {Copy Constraints, Do not copy springs, Do not copy Nodal Loads}.

The result of the operation is the following:



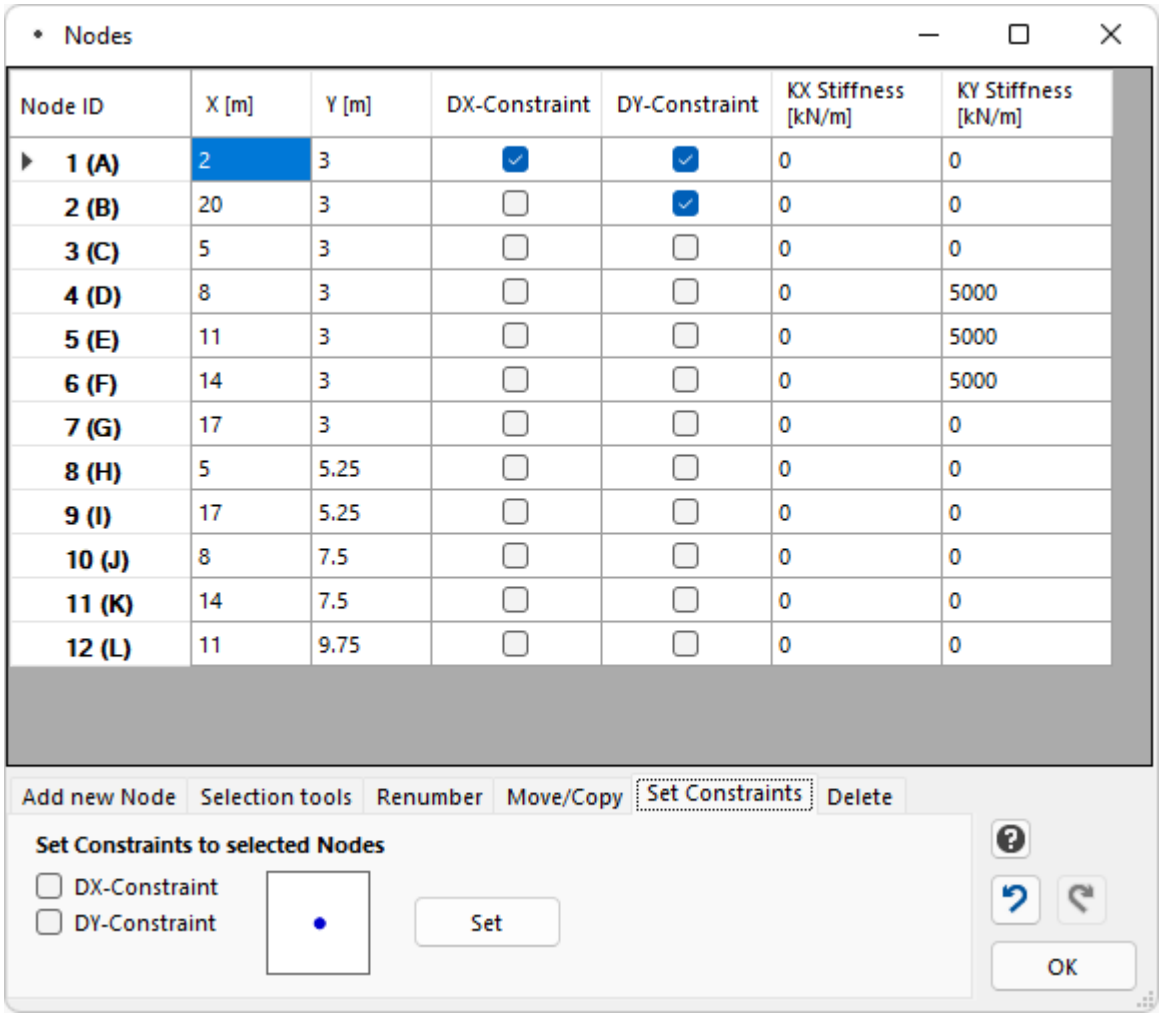
- Node D was already at (9, 3) and is not affected. New Node F is also created at the same location (9, 3).
- Constraints (and springs) are copied together with the Nodes.
- Nodal forces are NOT copied together with the Nodes, following the selected options.
- Copy Nodes does not affect any Elements as elements depend on existing Nodes only.

Set Constraints

In order to perform this action, **first you have to select the entire row(s) of the corresponding Node(s)** and then apply the action.

This tool is used for setting constraints on selected Nodes. You can set constraints for more than one Nodes at a time. First select the Nodes (rows) that you want to set the constraints on. Set the constraints by checking the DX and/or DY Constraint checkbox. Finally, click the "Set" button to perform the operation.

(c) 2022 EngiLab PC

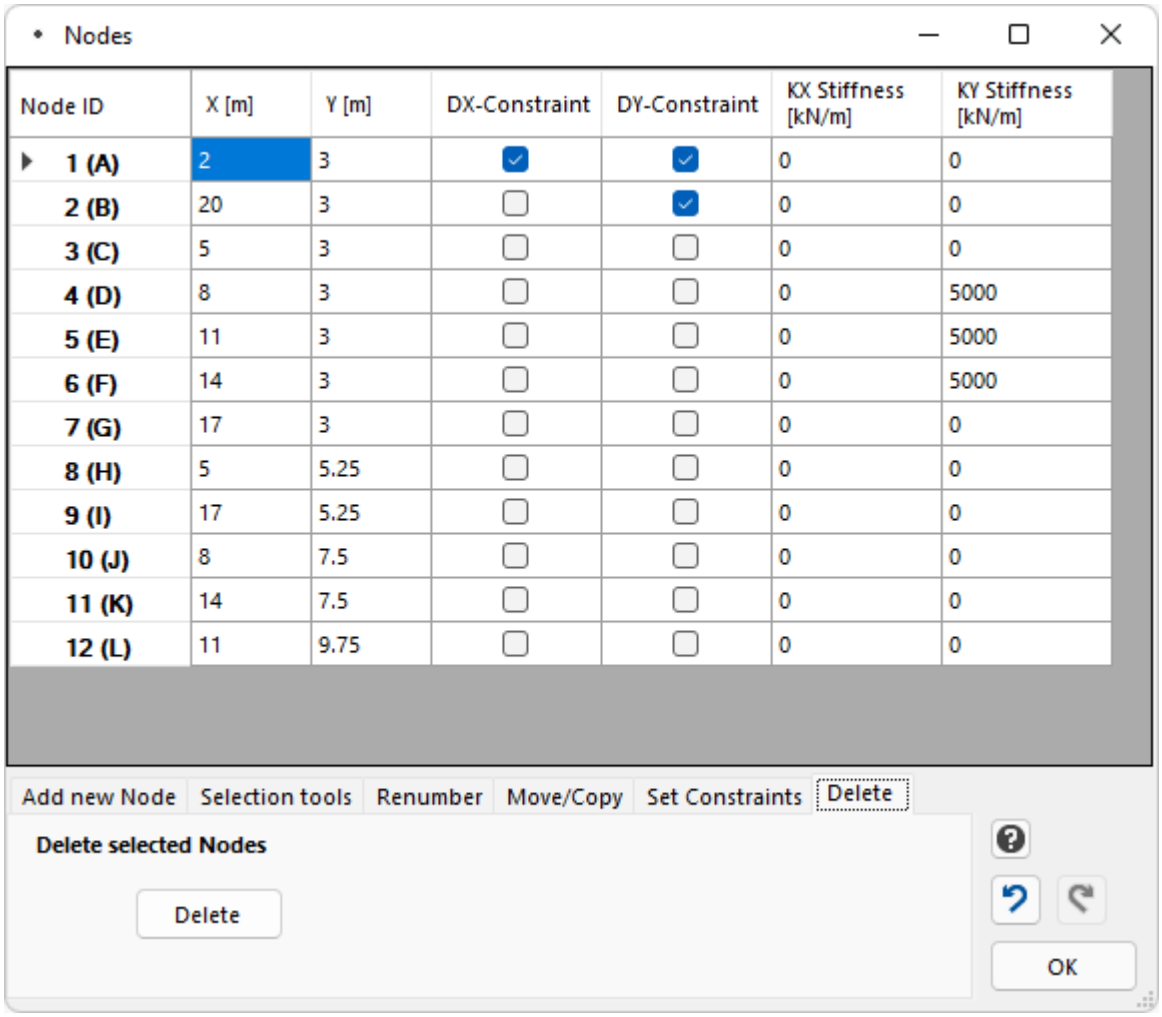


Delete Nodes

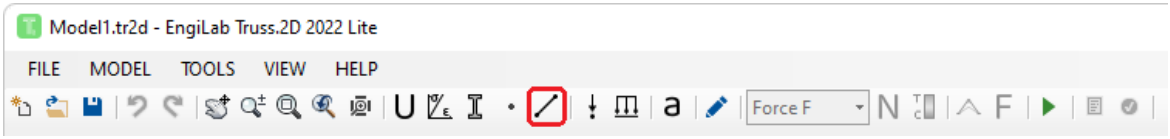
In order to perform this action, **first you have to select the entire row(s) of the corresponding Node(s)** and then apply the action.

This tool is used for deleting Nodes. You can Delete more than one Nodes at a time. First select the Nodes (rows) that you want to delete. Then, click the "Delete" button to delete the Nodes.

(c) 2022 EngiLab PC



2.2.5 Elements



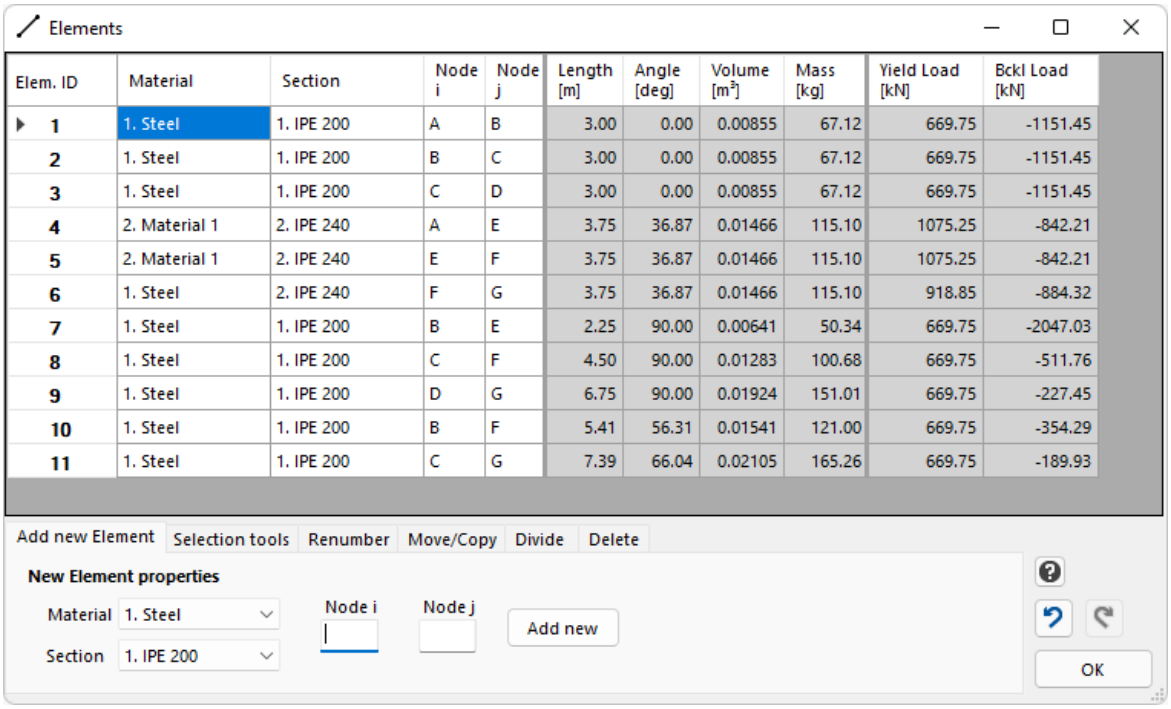
An Element connects two Nodes with each other. Each Element has to be assigned a Material and a Section. Before defining an Element, at least one Material and one Section have to be defined first.

(c) 2022 EngiLab PC

Element properties

The **Element** user-defined properties are the following:

- **Material ID**
- **Section ID**
- **Node i ID** (Start Node)
- **Node j ID** (End Node)



Elem. ID	Material	Section	Node i	Node j	Length [m]	Angle [deg]	Volume [m³]	Mass [kg]	Yield Load [kN]	Bckl Load [kN]
1	1. Steel	1. IPE 200	A	B	3.00	0.00	0.00855	67.12	669.75	-1151.45
2	1. Steel	1. IPE 200	B	C	3.00	0.00	0.00855	67.12	669.75	-1151.45
3	1. Steel	1. IPE 200	C	D	3.00	0.00	0.00855	67.12	669.75	-1151.45
4	2. Material 1	2. IPE 240	A	E	3.75	36.87	0.01466	115.10	1075.25	-842.21
5	2. Material 1	2. IPE 240	E	F	3.75	36.87	0.01466	115.10	1075.25	-842.21
6	1. Steel	2. IPE 240	F	G	3.75	36.87	0.01466	115.10	918.85	-884.32
7	1. Steel	1. IPE 200	B	E	2.25	90.00	0.00641	50.34	669.75	-2047.03
8	1. Steel	1. IPE 200	C	F	4.50	90.00	0.01283	100.68	669.75	-511.76
9	1. Steel	1. IPE 200	D	G	6.75	90.00	0.01924	151.01	669.75	-227.45
10	1. Steel	1. IPE 200	B	F	5.41	56.31	0.01541	121.00	669.75	-354.29
11	1. Steel	1. IPE 200	C	G	7.39	66.04	0.02105	165.26	669.75	-189.93

The derived Element properties (**Read-only properties**, calculated automatically by the program) are the following:

- **Length**: Length of the Element (in Length units, for example m). It is calculated based on the locations of the Start (i) and End (j) Nodes, as $L=\text{sqrt}[(Xj-Xi)^2+(Yj-Yi)^2]$.
- **Angle:** Orientation Angle of the Element (in Angle units, for example deg), measured counter-clockwise from the horizontal. When the Angle is zero, the Element points to the right (+X Global axis).
- **Volume**: Volume of the Element (in Volume units, for example m^3). It is calculated as the Length of the Element times the Area of the Cross section, V=L*A.
- **Mass**: Mass of the Element (in Mass units, for example kg). It is calculated as the Volume of the Element times the Density of the Material, M=V*d.
- **Yield Load**: Yield Load of the Element (in Stress units, for example MPa). It is calculated as the Yield Stress of the Material times the Area of the Cross section, Fy=fy*A.

(c) 2022 EngiLab PC

- **Buckling Load**: Euler's Critical Buckling Load (in Force units, for example kN). It is calculated as BL = $-\pi^2*E*I/L^2$, where I is the Moment of Inertia of the Section. BL is always reported with a negative sign (compression), as it is the compressive load at which a slender member will suddenly buckle.

Add new Element

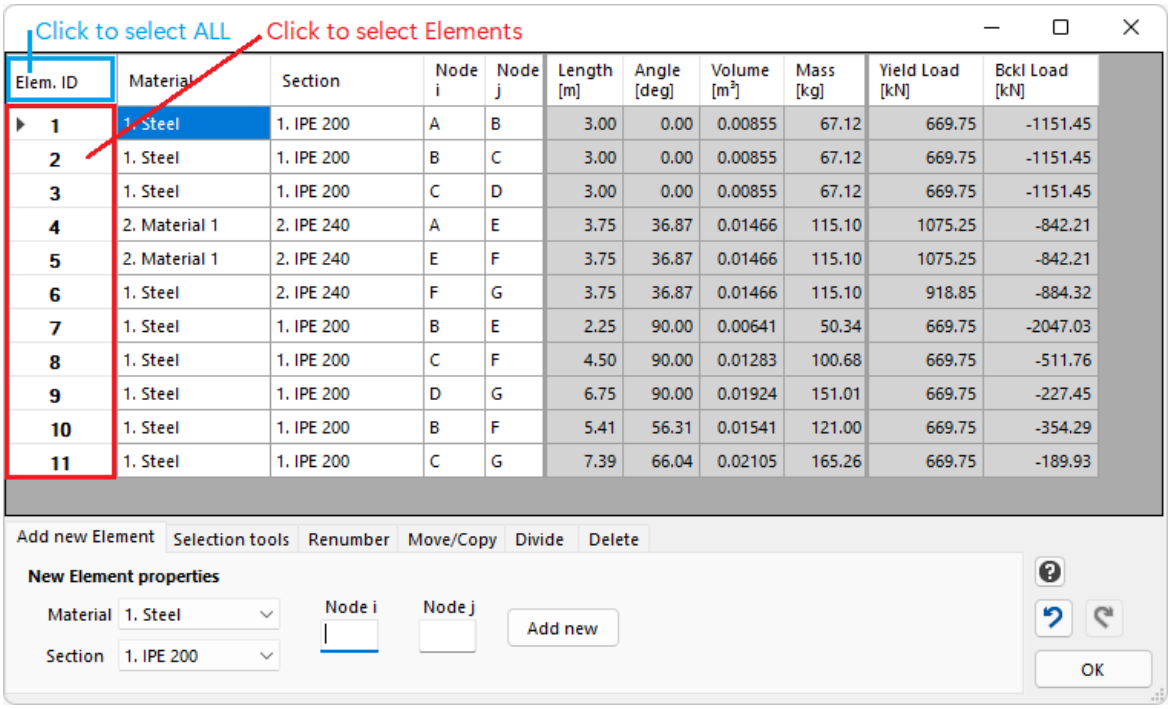
To add a new Element, select the Material and the Section and type the Start and End Nodes of the new Element. Then click the "Add new" button.

Edit an existing Element

You can click on the table and you can easily edit the properties of an already existing Element. Any change you make is automatically reflected to the Model on the screen.

Select existing Element(s)

You can click on a row number to select one or more Elements. Then you can apply actions to the selected Elements (renumber, move/copy, divide, delete, etc).

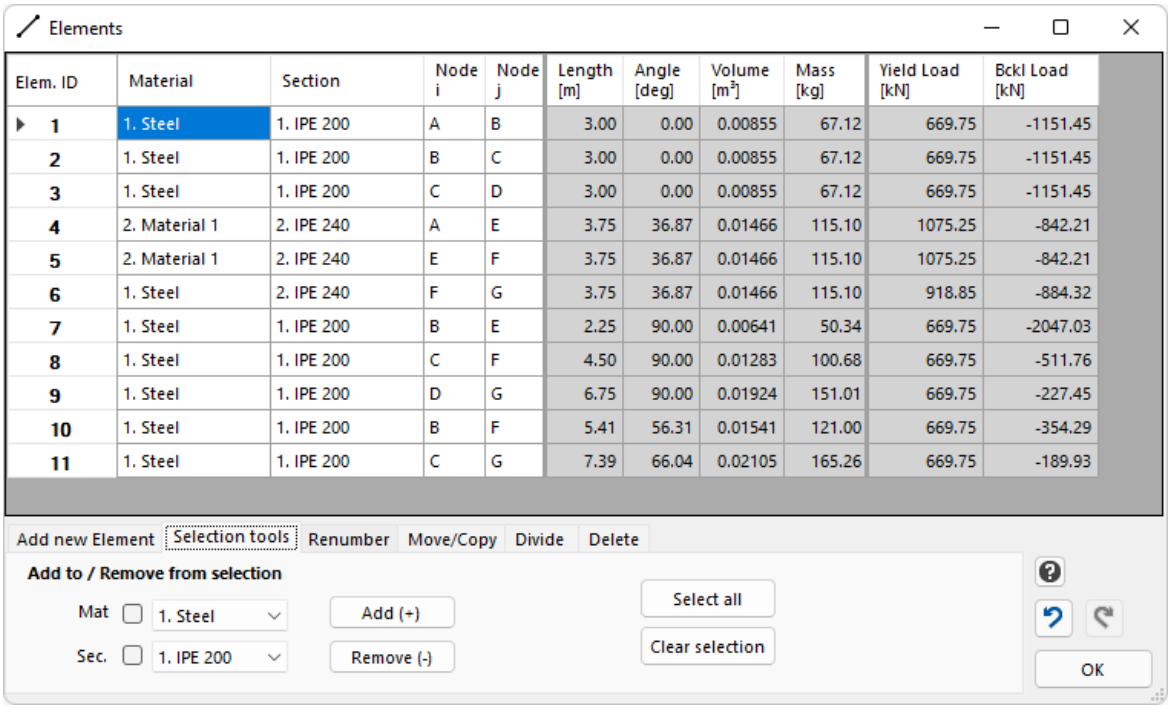


Selection tools

This tab provides tools for the selection of Elements. For example, you can select (add to selection) or deselect (remove from selection) all the Elements that are assigned a

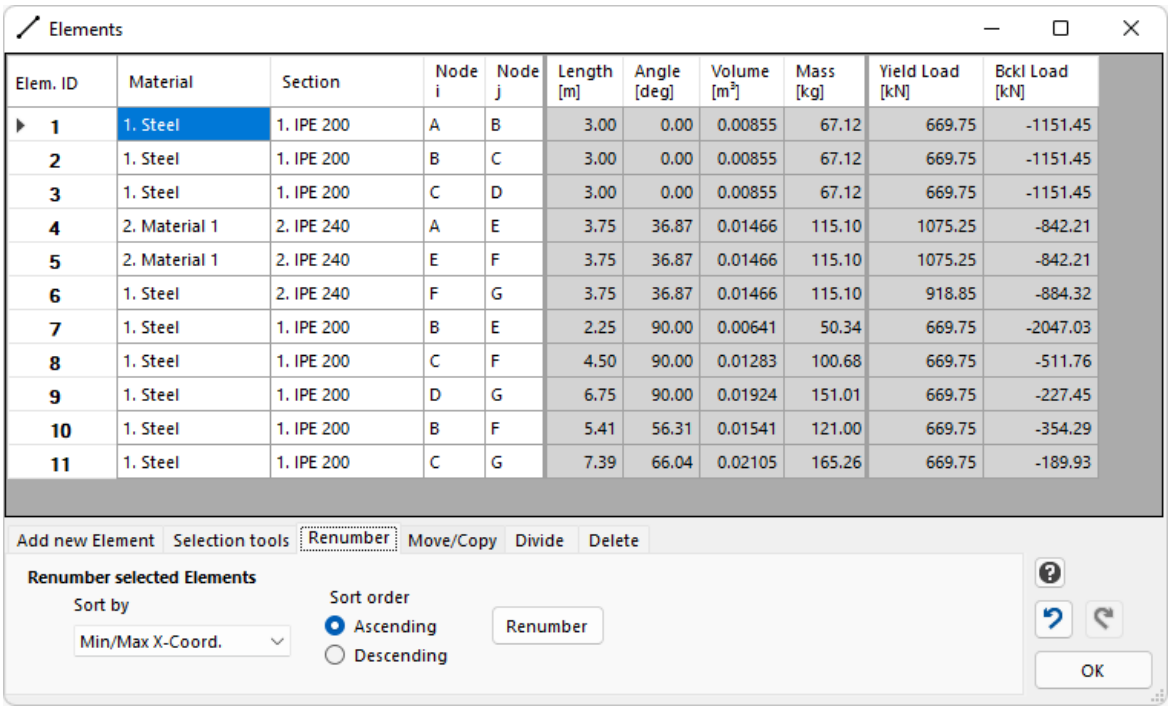
(c) 2022 EngiLab PC

specific Material and/or a specific Section. This is useful when for example you need to Delete all the Elements with a specific Section. You can also Select all Elements, or Clear the selection.



Renumber Elements

In order to perform this action, **first you have to select the entire row(s) of the corresponding Element(s)** and then apply the action.



This tool is for the renumbering of (the selected) Elements, based on various criteria:

- "**Min/Max X-Coord.**": If the sort order is "Ascending", then the Min X-Coordinate of the Element is used. If the sort order is "Descending", then the Max X-Coordinate is used.
- "**Min/Max Y-Coord.**": If the sort order is "Ascending", then the Min Y-Coordinate of the Element is used. If the sort order is "Descending", then the Max Y-Coordinate is used.
- "**Min/Max Node ID**": If the sort order is "Ascending", then the Min Node ID of the Element is used. If the sort order is "Descending", then the Max Node ID is used.
- "**Start Node (i) ID**"
- "**Start Node (i) X-Coord.**"
- "**Start Node (i) Y-Coord.**"
- "**End Node (j) ID**"
- "**End Node (j) X-Coord.**"
- "**End Node (j) Y-Coord.**"

First, the Elements have to be selected. Then we select the "Sort by" field and then the sort order (ascending or descending). Finally, we click the "Renumber" button.

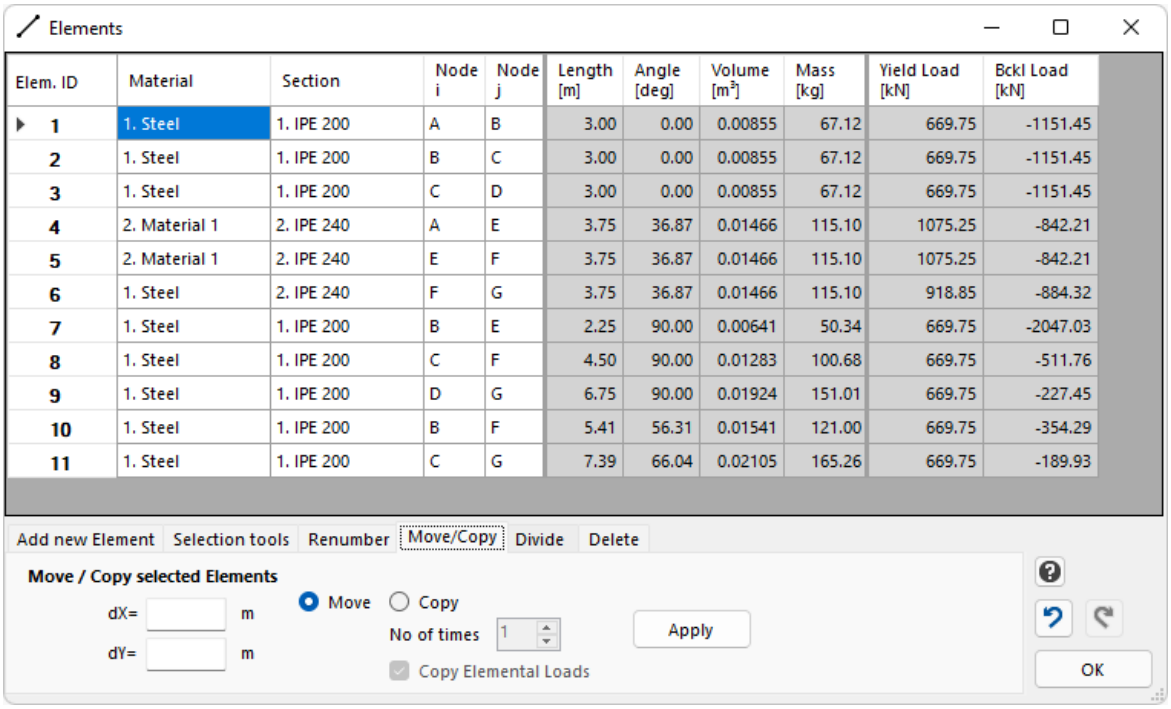
(c) 2022 EngiLab PC

Important: Renumbering is done for the selected Elements. The non-selected Elements remain unchanged. To renumber all Elements, all Elements have to be selected first.

Move/Copy Elements

In order to perform this action, **first you have to select the entire row(s) of the corresponding Element(s)** and then apply the action.

This tool is used for moving or copying Elements. The user specifies the Move/Copy Vector {V}={dX, dY}.



Move Elements

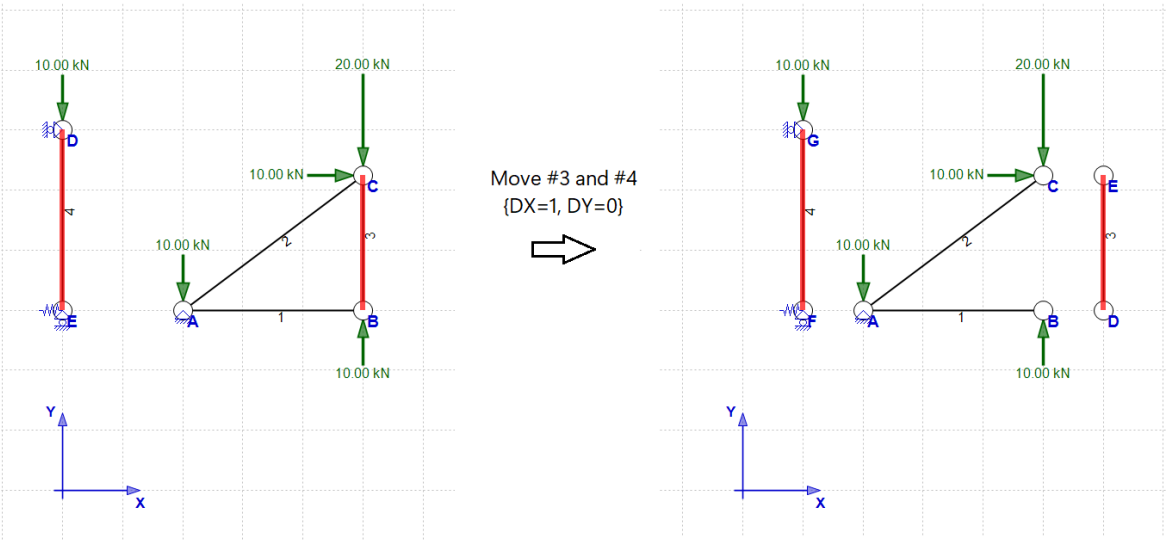
- Multiple Elements (one or more) can be selected to be moved at once.
- Move Elements can only be applied once (only one time), not a number of times (only copy works with the option to apply it over a number of times).
- With the Move command, several Elements can be moved together to new locations. The user defines the move vector as {dX, dY}.
- Elements are moved together with any Elemental Loads on them.
- If an Element is moved that is not connected to other Elements, then its start and end nodes are actually moved (**see Example 1** below), together with any Constraints, Springs and Nodal Loads on them. As a result, the element moves.

(c) 2022 EngiLab PC

- If an Element is moved that was previously connected to other Elements via common Nodes (Elements that are not selected to move, also), then the Element is detached from the other Elements and new Nodes are generated to define its new position. The previous Nodes stay connected with the other Elements, together with any Constraints, Nodal Loads or Springs they had (**see Example 1** below).
- If an Element is moved and there is already an existing Node in the new position (either at the start i or at the end j of the new position of the Element, or at both ends i and j), then the existing Node is used for the definition of the Element, i.e. the Element is connected to the existing Node (**see Example 2** below).

Example 1 - Moving Elements, detaching Elements from Nodes

In the example below (left), Elements 3 and 4 are moved with the vector {DX=1, DY=0}. The result of the operation is shown on the right.



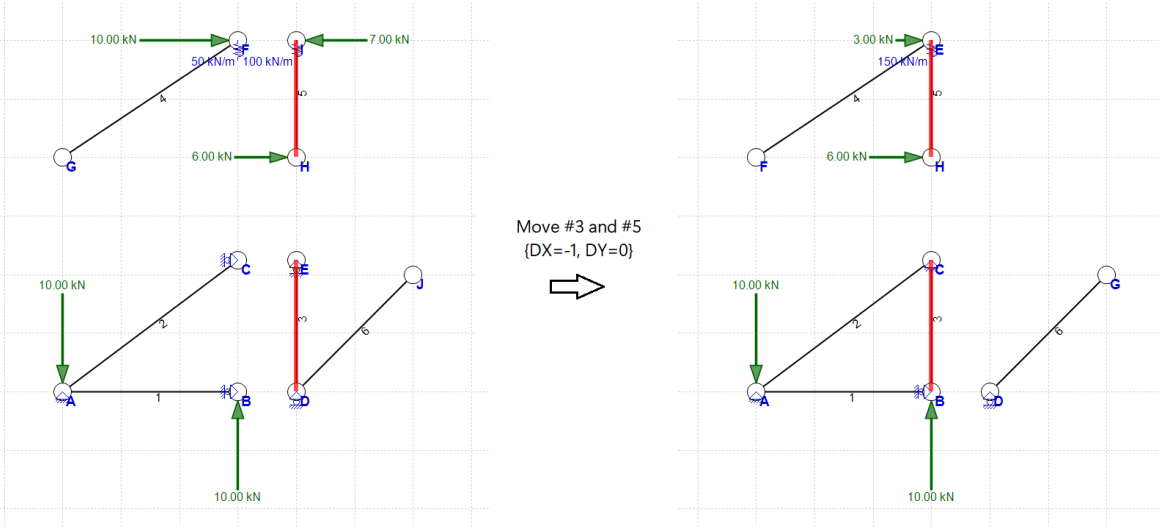
Note the following:

- Element 4 is moved together with its Nodes.
- Element 3 is detached from Elements 1 and 2 and it is now on its own, not connected to other Elements.
- 2 new Nodes have been automatically generated for this (Nodes D and E in the model on the right).
- Some Node renumbering has occurred as a result of the operation. Element 4 connected Nodes D and E before, while after the operation it connects Nodes F and G. You can use the Nodes renumbering tool to renumber Nodes, if needed.
- The Nodal Load on Node D is moved together with the Node (numbered as Node G after the operation). The Nodal Loads on Nodes B and C stay with the old Nodes.

(c) 2022 EngiLab PC

Example 2 - Moving Elements, attaching Elements to Nodes

In the example below (left), Elements 3 and 5 are moved with the vector {DX=-1, DY=0}. The result of the operation is shown on the right.



- After the move operation, the end Node (I) of Element 5 is merged with the existing Node F and the Element is attached (connected) to Node F. As a result, Element 5 is now connected to Element 4. Nodal loads and springs are combined in this case. The new Nodal Load on Node E (new numbering) is 10-7=3, while the new spring stiffness is 50+100=150.
- Element 3 is detached from Node D. And now Element 3 is connected to Nodes B and C. The two supports are combined on the top end of the element. Node C now has a pinned support, while the previous supports were two rollers that are now combined. Supports are not combined at the bottom end of Element 3, as the roller support of Node D stays behind, together with the Node which is still needed for the definition of Element 6.

Copy Elements

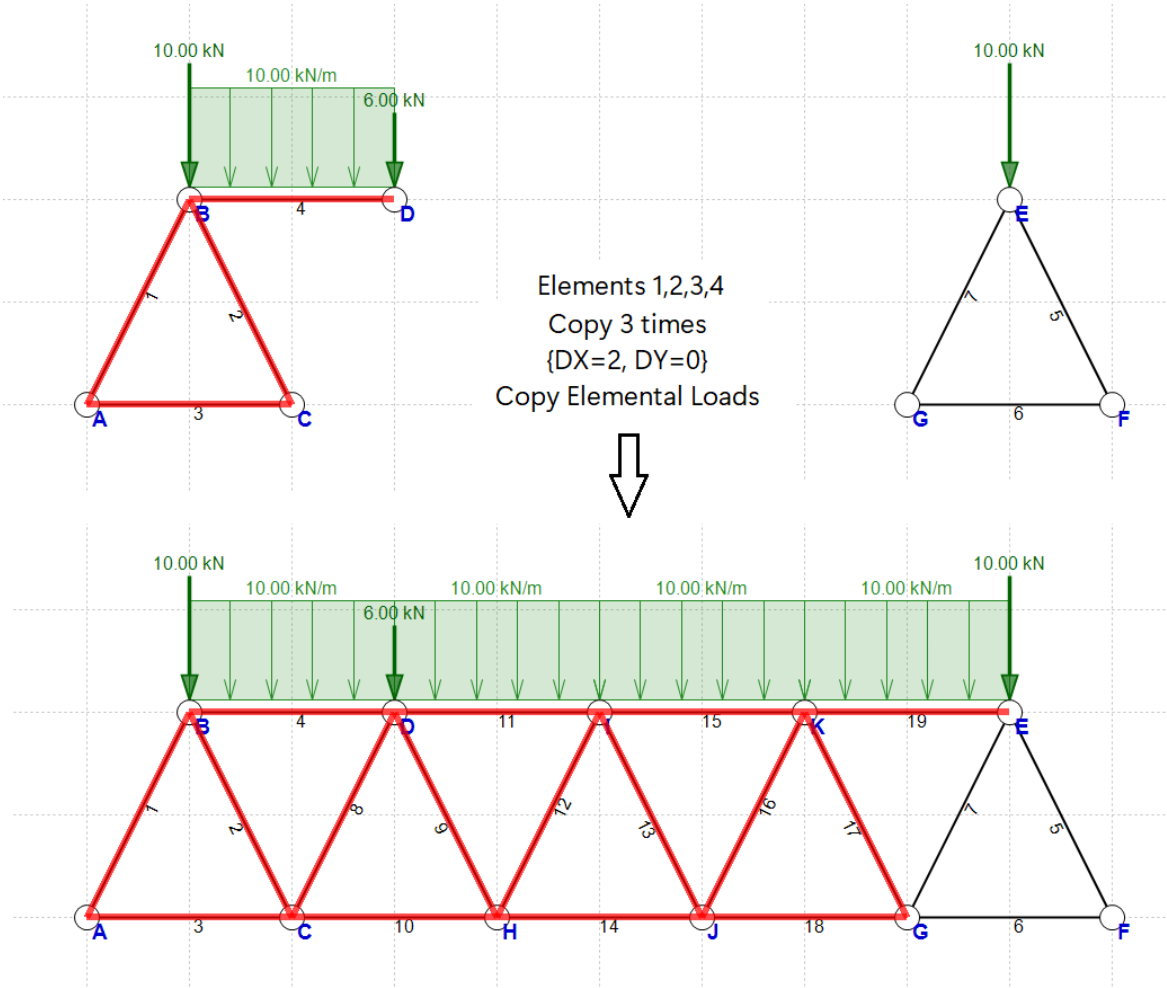
- Multiple Elements (one or more) can be selected to be copied at once.
- Copy Elements can be applied multiple times (up to 50) with a single click.
- With the Copy command, several Elements can be copied to new locations. The user defines the copy vector as {dX, dY} and the number of times the operation will be applied.

(c) 2022 EngiLab PC

- If the check box "Copy Elemental Loads" is checked, Elements are copied together with their Elemental Loads.
- By default, the "Copy Elemental Loads" check box is checked. This can be changed by the user.

Example 3 – Copying Elements

Copy Elements works similarly with Move Elements, but this time new Elements are created instead of moving already existing ones. See the example below:



We now copy Elements 1, 2, 3 and 4 with the vector {DX=2, DY=0}, 3 times. The "Copy Elemental Loads" check box is checked. The result of the operation is shown in the figure:

- New Elements and Nodes are created and as a result we now have a fully connected Truss structure.

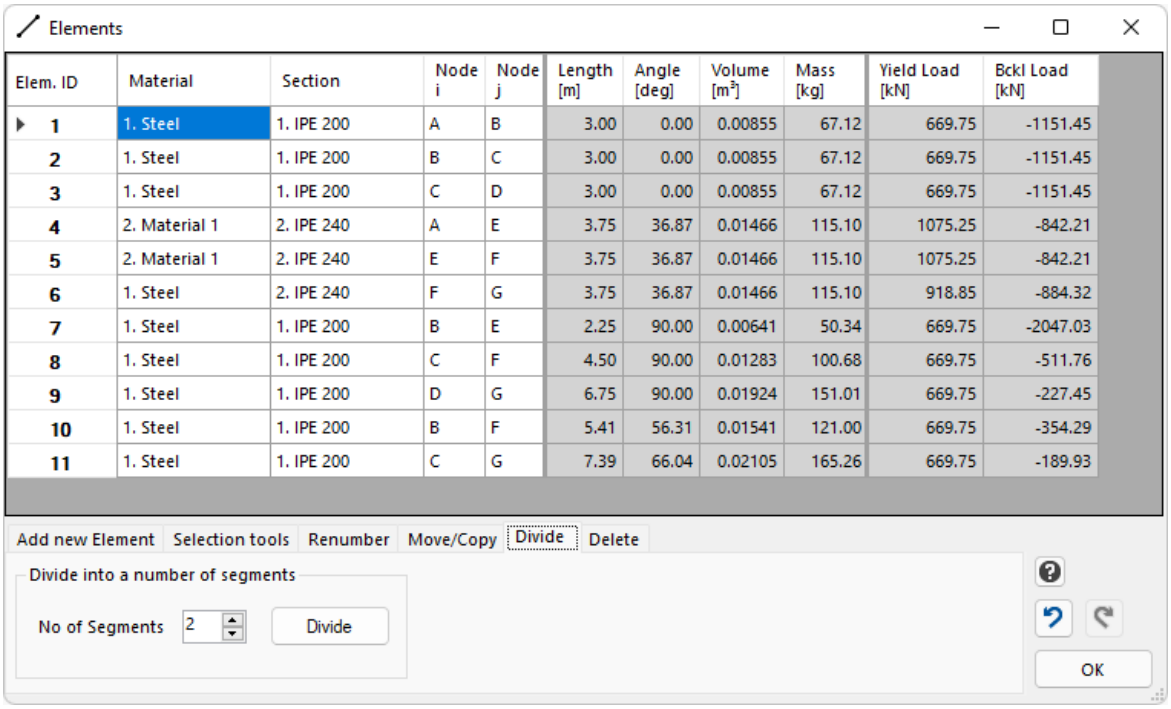
(c) 2022 EngiLab PC

- Node E was already at (9, 6) and it is used for the connection of Element 19. No new Node is created there, the old Node is instead used.
- Node G was already at (8, 4) and it is used for the connection of Elements 17 and 18. No new Node is created there, the old Node is instead used.
- Elemental Loads are copied together with the Elements, because the check box "Copy Elemental Loads" was checked.
- This techniques offers a very fast and efficient solution to create Trusses with repeated patterns.

Divide Elements

In order to perform this action, **first you have to select the entire row(s) of the corresponding Element(s)** and then apply the action.

Select the Elements (rows). Type the number of the division segments (up to 40) and then click the "Divide" button. You can Divide more than one Elements at a time.



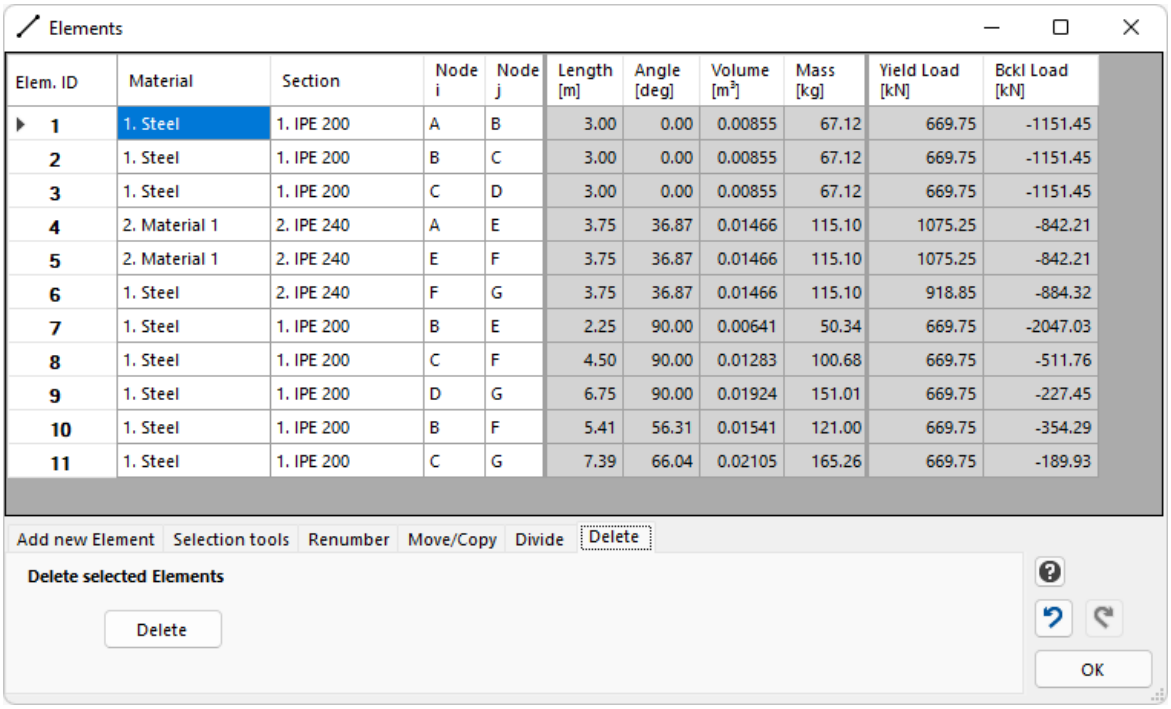
Elem. ID	Material	Section	Node i	Node j	Length [m]	Angle [deg]	Volume [m³]	Mass [kg]	Yield Load [kN]	Bckl Load [kN]
1	1. Steel	1. IPE 200	A	B	3.00	0.00	0.00855	67.12	669.75	-1151.45
2	1. Steel	1. IPE 200	B	C	3.00	0.00	0.00855	67.12	669.75	-1151.45
3	1. Steel	1. IPE 200	C	D	3.00	0.00	0.00855	67.12	669.75	-1151.45
4	2. Material 1	2. IPE 240	A	E	3.75	36.87	0.01466	115.10	1075.25	-842.21
5	2. Material 1	2. IPE 240	E	F	3.75	36.87	0.01466	115.10	1075.25	-842.21
6	1. Steel	2. IPE 240	F	G	3.75	36.87	0.01466	115.10	918.85	-884.32
7	1. Steel	1. IPE 200	B	E	2.25	90.00	0.00641	50.34	669.75	-2047.03
8	1. Steel	1. IPE 200	C	F	4.50	90.00	0.01283	100.68	669.75	-511.76
9	1. Steel	1. IPE 200	D	G	6.75	90.00	0.01924	151.01	669.75	-227.45
10	1. Steel	1. IPE 200	B	F	5.41	56.31	0.01541	121.00	669.75	-354.29
11	1. Steel	1. IPE 200	C	G	7.39	66.04	0.02105	165.26	669.75	-189.93

Delete Elements

(c) 2022 EngiLab PC

In order to perform this action, **first you have to select the entire row(s) of the corresponding Element(s)** and then apply the action.

This tool is used for deleting Elements. First select the Elements (rows) that you want to delete. Then, click the "Delete" button to delete the Elements. You can Delete more than one Elements at a time.



Elem. ID	Material	Section	Node i	Node j	Length [m]	Angle [deg]	Volume [m³]	Mass [kg]	Yield Load [kN]	Bckl Load [kN]
1	1. Steel	1. IPE 200	A	B	3.00	0.00	0.00855	67.12	669.75	-1151.45
2	1. Steel	1. IPE 200	B	C	3.00	0.00	0.00855	67.12	669.75	-1151.45
3	1. Steel	1. IPE 200	C	D	3.00	0.00	0.00855	67.12	669.75	-1151.45
4	2. Material 1	2. IPE 240	A	E	3.75	36.87	0.01466	115.10	1075.25	-842.21
5	2. Material 1	2. IPE 240	E	F	3.75	36.87	0.01466	115.10	1075.25	-842.21
6	1. Steel	2. IPE 240	F	G	3.75	36.87	0.01466	115.10	918.85	-884.32
7	1. Steel	1. IPE 200	B	E	2.25	90.00	0.00641	50.34	669.75	-2047.03
8	1. Steel	1. IPE 200	C	F	4.50	90.00	0.01283	100.68	669.75	-511.76
9	1. Steel	1. IPE 200	D	G	6.75	90.00	0.01924	151.01	669.75	-227.45
10	1. Steel	1. IPE 200	B	F	5.41	56.31	0.01541	121.00	669.75	-354.29
11	1. Steel	1. IPE 200	C	G	7.39	66.04	0.02105	165.26	669.75	-189.93

2.2.6 Nodal Loads



Point loads can only be NODAL loads (acting on Nodes). To define a load (FX and/or FY) at a specific point, first a Node must have been defined at that location.

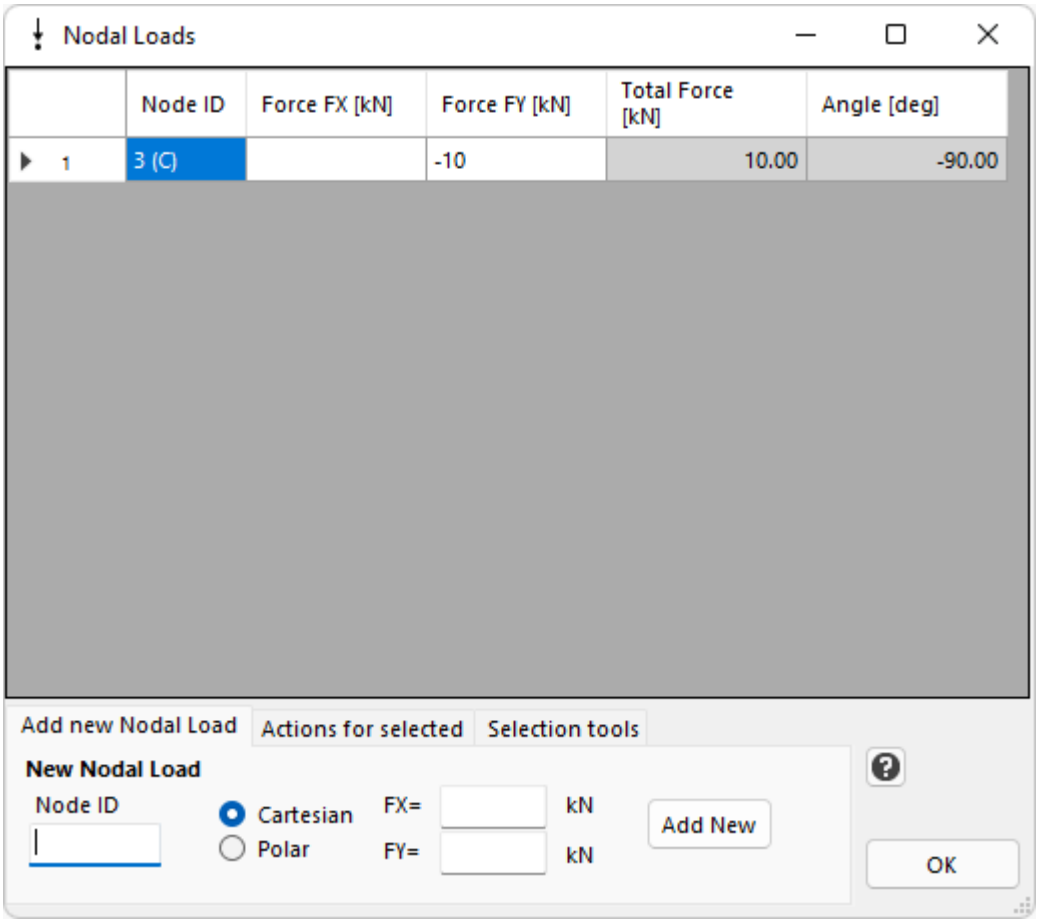
The **Nodal Load** properties are the Loads in each Degree Of Freedom (DOF) of the Node (Sign convention: Global axes):

- **Force FX**, positive when pointing to the right
- **Force FY**, positive when pointing up

(c) 2022 EngiLab PC

The derived Nodal Load properties (**Read-only properties**, calculated automatically by the program) are the following:

- **Total Force**: The magnitude of the Total Force acting on the Node (always positive), which is calculated as TotalForce=sqrt(FX^2+FY^2)
- **Angle**: The Angle defining the orientation of the Total Force vector. Angles are measures counter-clockwise from the horizontal: Angle 0 means that the Total Force points to the right (positive X-Global Axis). Angle 90 deg means that the Total Force points upwards (positive Y-Global Axis), etc. Angle -90 deg means that the Total Force points downwards (negative Y-Global Axis), etc.



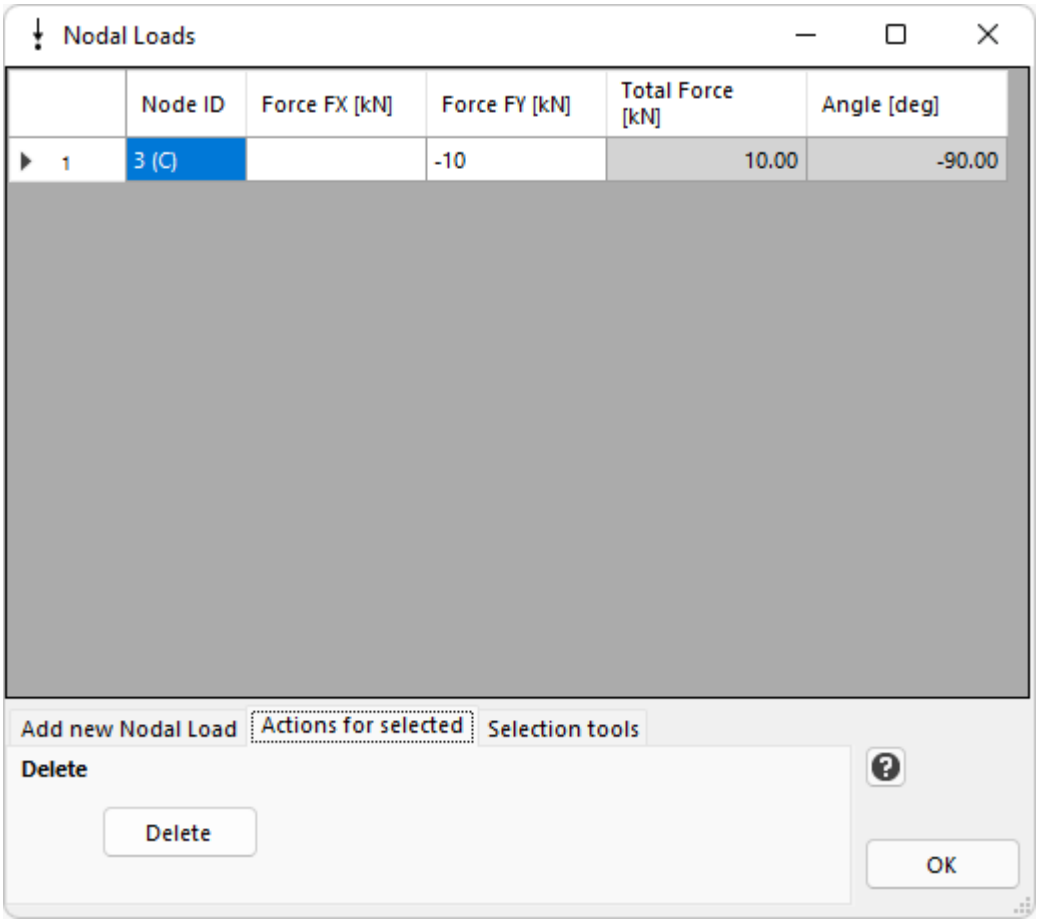
Add new Nodal Load

To add a new Nodal Load, specify the Node ID and then type the FX and FY values of the Nodal Load set. Then click the "Add Nodal Load" button.

Actions for selected Nodal Loads

In order to perform these actions, **you have to first select the entire row(s) of the corresponding Nodal Load(s)** and then apply the action.

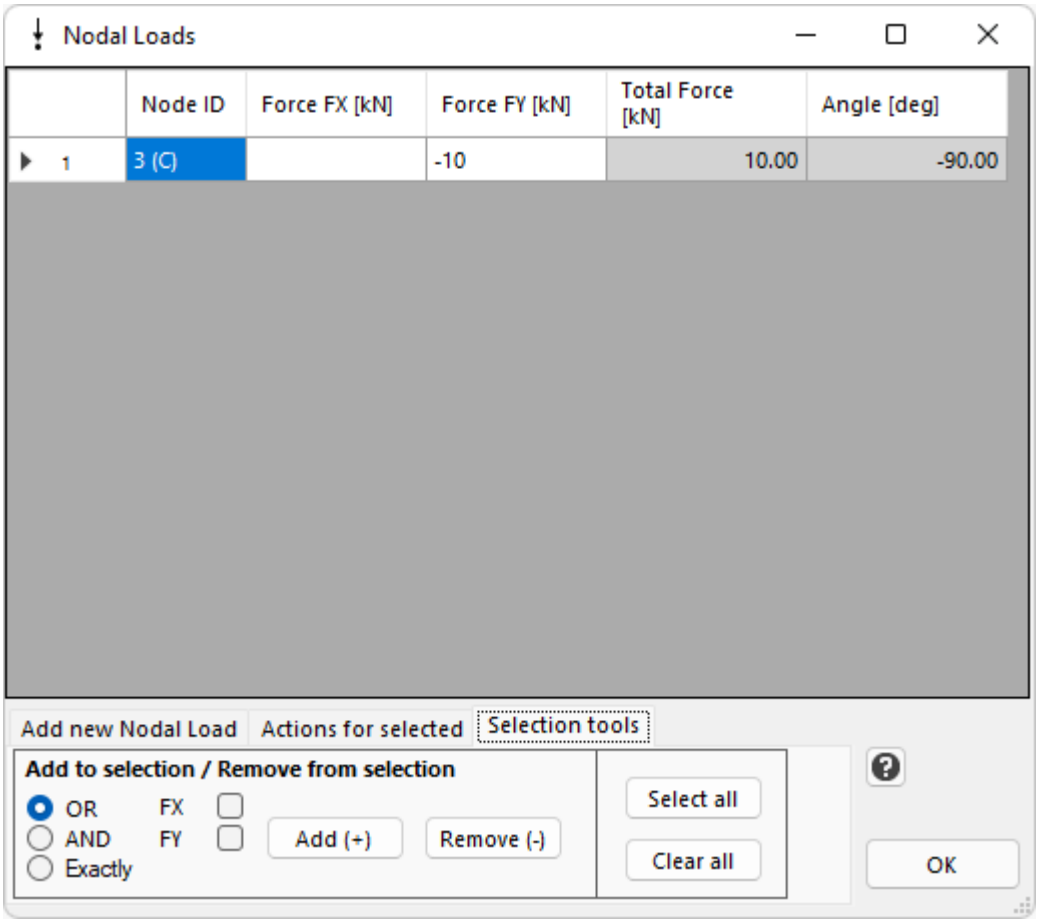
- **Delete Nodal Load(s)**. Click the "Delete" button. You can Delete more than one Nodal Loads at a time.



(c) 2022 EngiLab PC

Nodal Loads selection tools

This tab provides tools for the selection of Nodal Loads. For example, you can select (add to selection) or deselect (remove from selection) all the Nodal Loads that have FX and/or FY values.



2.2.7 Elemental Loads



The **Elemental Load** properties are the loads acting at the X and/or Y directions (**in Global axes system**) along an Element. EngiLab Truss.2D supports uniform loads along Elements. The Elemental Loads are applied along the Element and must be given as Force per Unit Length of the element.

See Important Note about the Elemental Loads on a 2D Truss structure, at the end of the page.

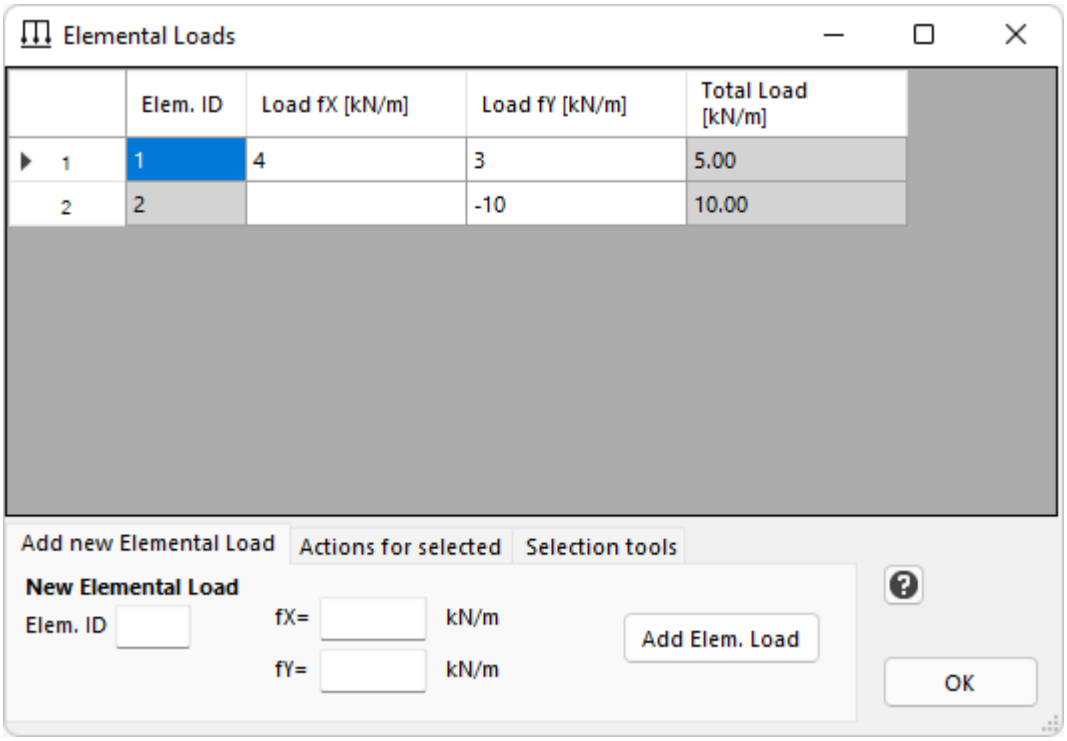
(c) 2022 EngiLab PC

The Elemental Load properties are the following:

- **Force fX**: X-Uniform Load acting along the element, positive when pointing to the right
- **Force fYi**: Y-Uniform Load acting along the element, positive when pointing upwards

The derived Elemental Load properties (**Read-only properties**, calculated automatically by the program) are the following:

- **Total Load**: The magnitude of the Total Load acting on the Element (always positive), which is calculated as TotalLoad=sqrt(fX^2+fY^2)



Add new Elemental Load

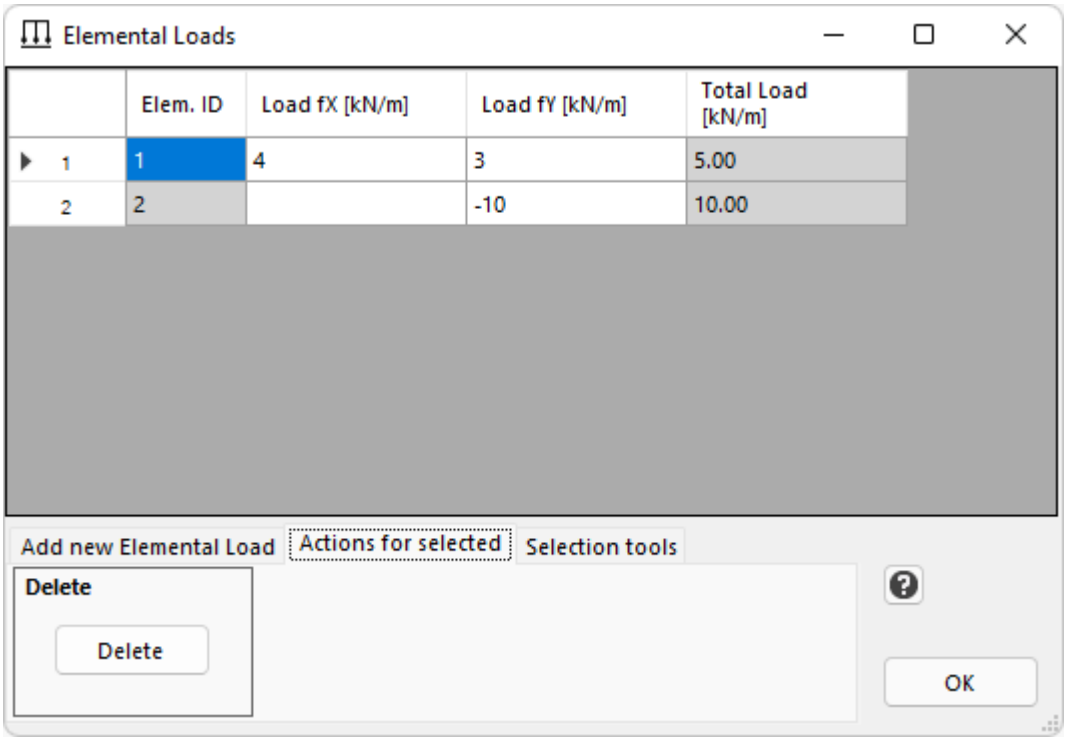
To add a new Elemental Load, first specify the Element ID. Then specify the fX and fY values and click the "Add Elem. Load" button.

(c) 2022 EngiLab PC

Actions for selected Elemental Loads

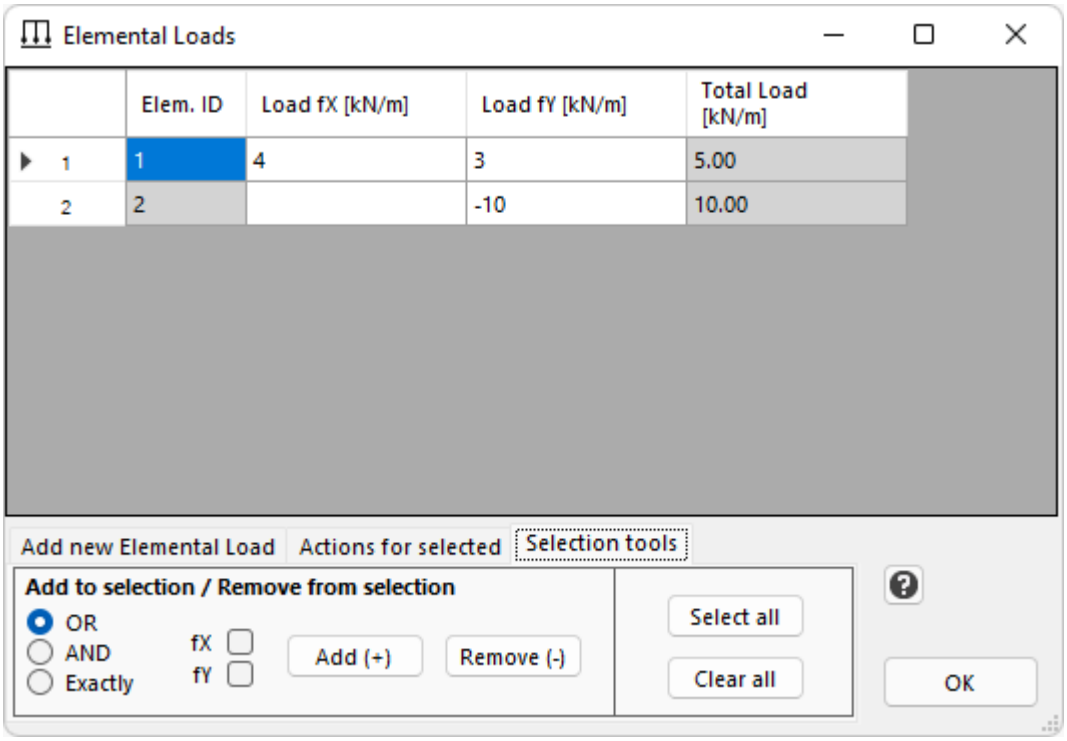
In order to perform these actions, **you have to first select the entire row(s) of the corresponding Elemental Load(s)** and then apply the action.

- **Delete Elemental Load(s)**. Click the "Delete" button. You can Delete more than one Elemental Loads at a time.



Elemental Loads selection tools

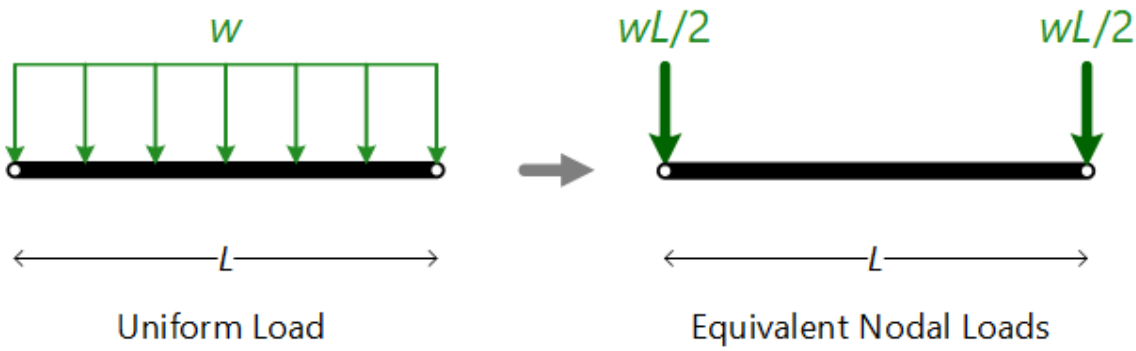
This tab provides tools for the selection of Elemental Loads. For example, you can select (add to selection) or deselect (remove from selection) all the Elemental Loads that have fX and/or fY values.



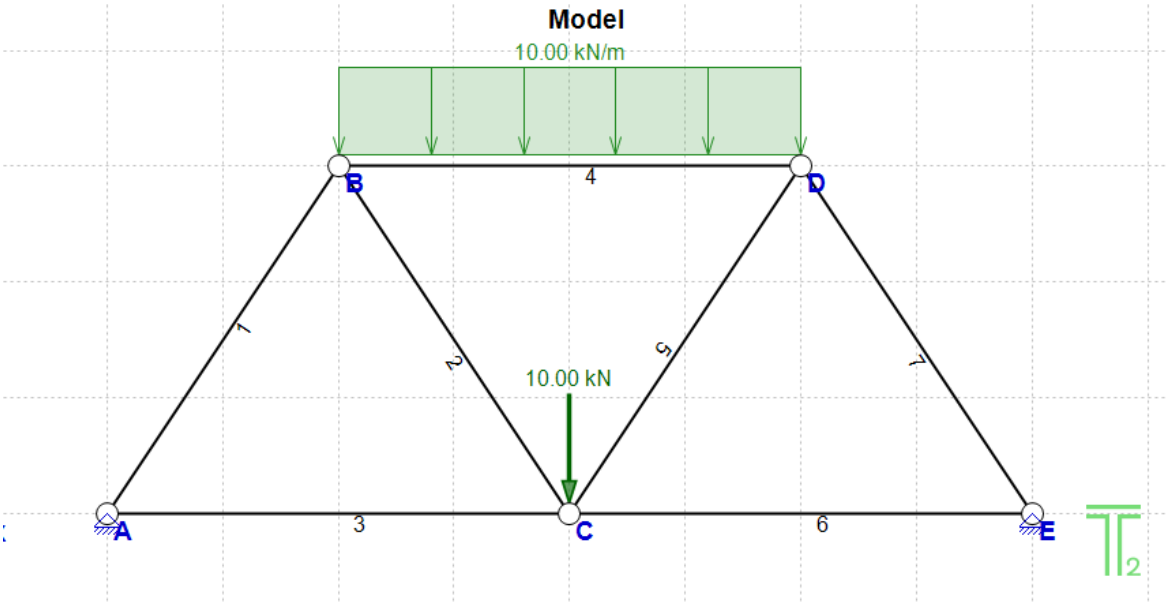
Important Note

In a 2D Truss, ALL loads are applied at the joints (Nodes in our case). Truss members are not supposed to bend and they have only axial force. No shear force and no bending moment exist on a Truss Element. In EngiLab Truss.2D, the user can specify Elemental Loads along an Element, but these loads are not really handled as Elemental Loads. They are automatically transferred to the corresponding Nodes of the Element, and they eventually become joint (Nodal) Loads for the Modal. This way the Elemental loads can be applied, but they act as joint (Nodal) Loads for the Analysis. This is shown schematically below.

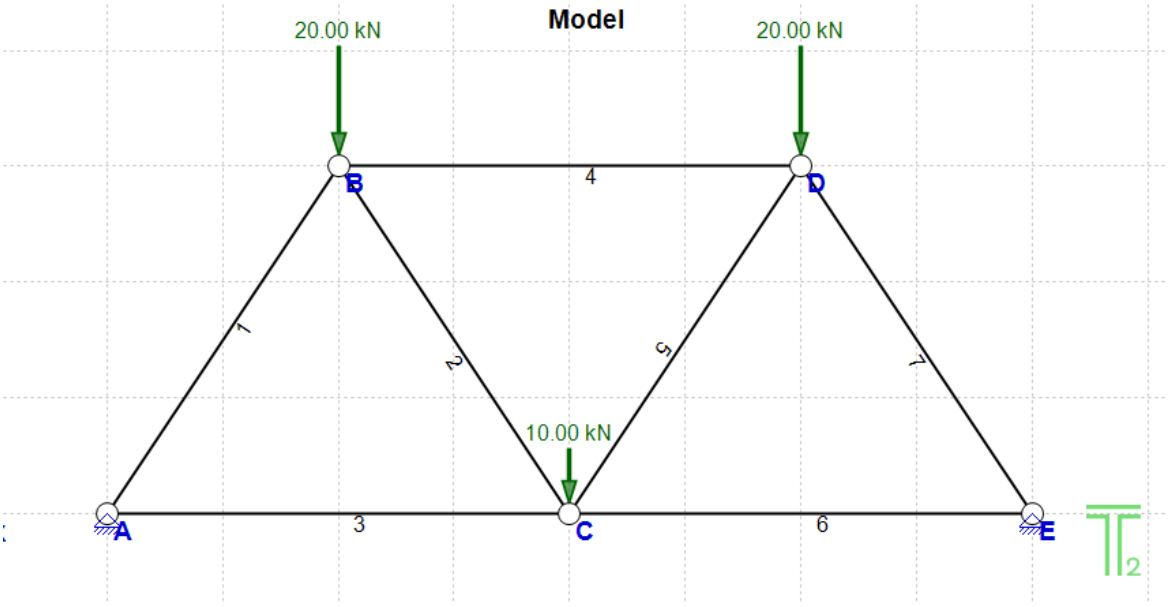
(c) 2022 EngiLab PC



See also the example below:



The equivalent Model that is analyzed by the program is the following, where the load of 10*4=40 kN has been transferred to the two Nodes (B and D). Each Node takes half of the load.



The above two Models behave exactly the same and they will give exactly the same Analysis results. The same approach is followed with Acceleration (Body) Loads which are also uniform loads and are transferred to the Nodes using the same mechanism.

2.2.8 Body (Acceleration) Loads



You can specify the **X and Y components** of an Acceleration vector acting on the Model. If the Model has Mass (Non-zero Density is defined for the Model's Material), then the acceleration will cause inertial body (Elemental) forces on the Elements of the Model, according to Newton's Second Law of Motion. These elemental forces are calculated automatically by the program and the results are shown on the form.

To apply an acceleration vector, type the values for the aX and aY acceleration components and **press the ENTER on the keyboard**. The model is updated automatically. The values of the X- and Y- Body Loads can be seen on the form (as read-only properties), together with the Material Density, the Section Area and the Length of each member.

The calculation is the following:

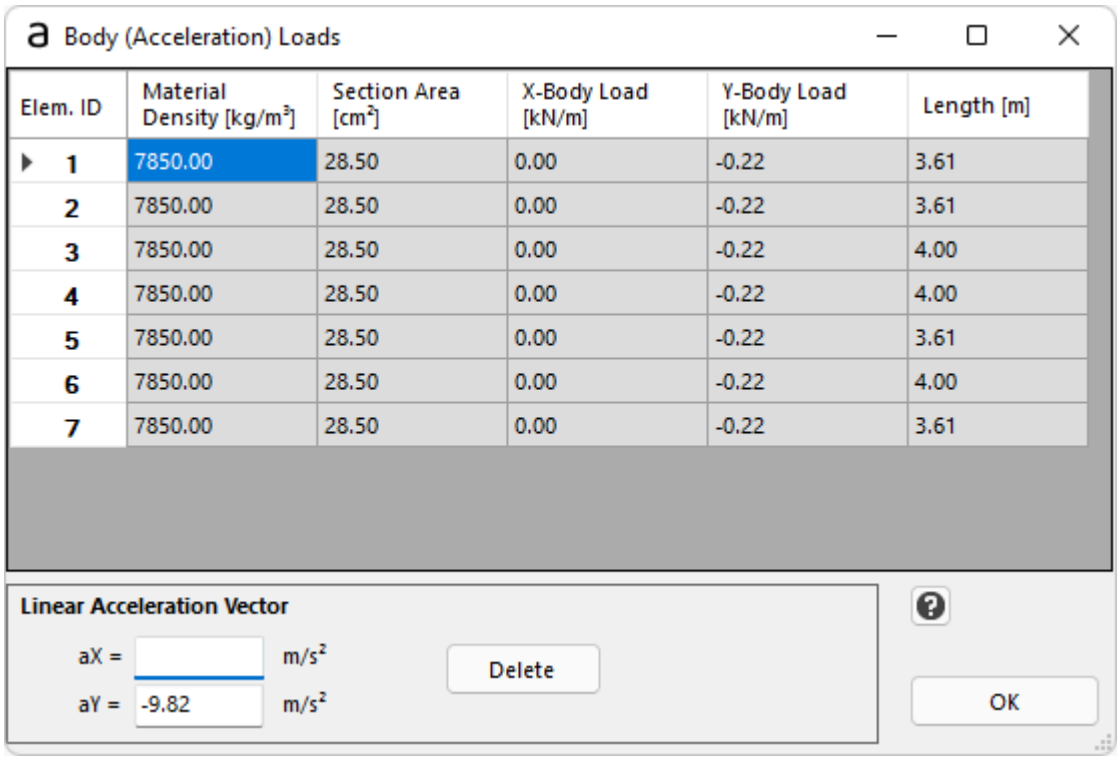
The total force on an element due to the Acceleration is: $F=m*a$ (Newton's Second Law of Motion) = $d*V * a = d*A*L * a$,

where m is the mass, a is the acceleration, d is the density, V is the volume and L is the length of the Element.

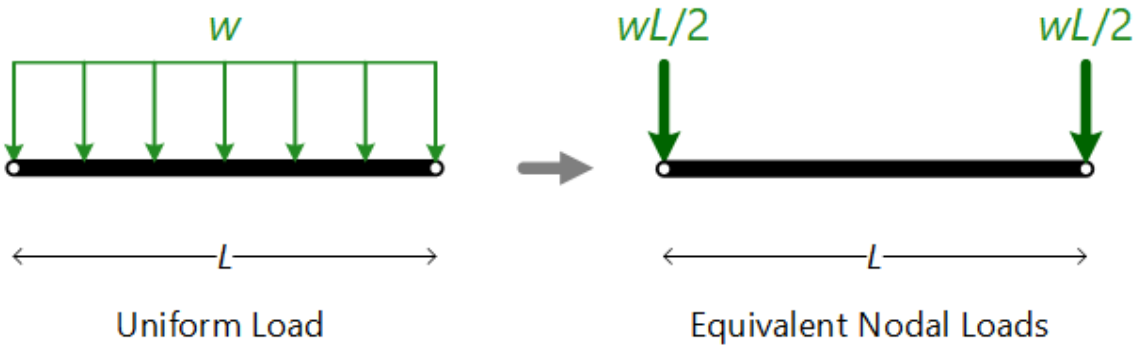
Accordingly, the uniform Load (per unit length of the Element) is: $f=F/L = d*A * a$.

(c) 2022 EngiLab PC

As a result, the uniform load applied on the Element due to an acceleration vector (in X or Y direction) is finally ***f=d*A*a***



Important: In EngiLab Truss.2D all distributed loads are converted to equivalent Nodal Loads. In the end, for the Analysis needs, there will be only Nodal Loads on the structure. Doing this, the effects of bending on the Elements are neglected. The approach is shown schematically below.



Useful applications of body (Acceleration) loads

1. Define the self-weight of the Model or the self-weight of specific Elements

If you want to take into account the self-weight of Elements as an additional Elemental Load for each Element, then you have to provide the Material Density for the Material of each Element (See Materials) and also to define a Linear Acceleration Vector equal to the standard earth gravitational acceleration. A common practice is to put the earth gravitational acceleration with a minus (-) sign at the Y direction - this means gravity acting towards -Y global axis.

Example using Consistent Units: If the Consistent System of Units is used and you assume that you are using kN for forces, m for length and s for time, then the Material Density has to be given in t/m^3 and you have to input -9.80665 (or simply -9.81) at the aY component of the Linear Acceleration Vector. This way, the self-weight will be applied to all Elements of the Model that have Mass (i.e. are assigned a Material which has a Density value). Note that Density is an optional property of a Material. If there is no Density for the Material of an Element, then the self-weight of the Element will be considered as zero and there will be no body load.

Using another System of Units: If Metric, US or another specific System of Units is used, then the Acceleration can be given using any Acceleration unit, as defined in the Predefined Systems section.

2. Define forces acting on a moving (accelerating) object

Acceleration is not only Gravity. If an object accelerates, then inertial forces are applied on it, provided that the object has Mass. For example, you can analyze a structure that is fixed on a moving (accelerating) vehicle. Even if there are no other loads, the acceleration of the vehicle will cause inertial forces on the object. By providing the X and Y components of the acceleration vector, you can calculate these forces and analyze the Model.

(c) 2022 EngiLab PC

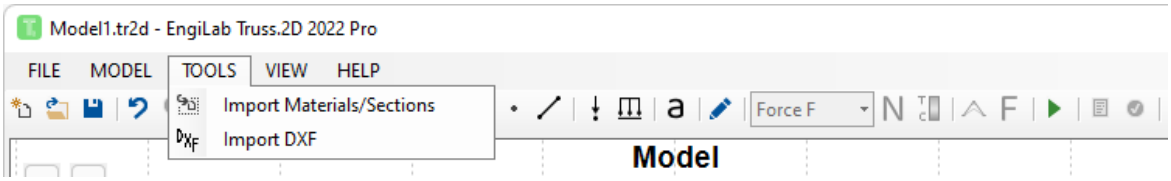
This page intentionally left blank.

Chapter



Tools

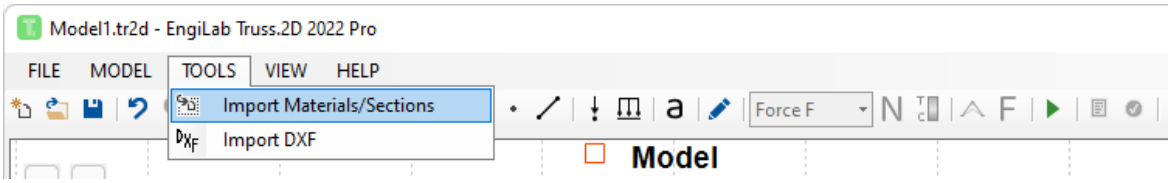
3 Tools



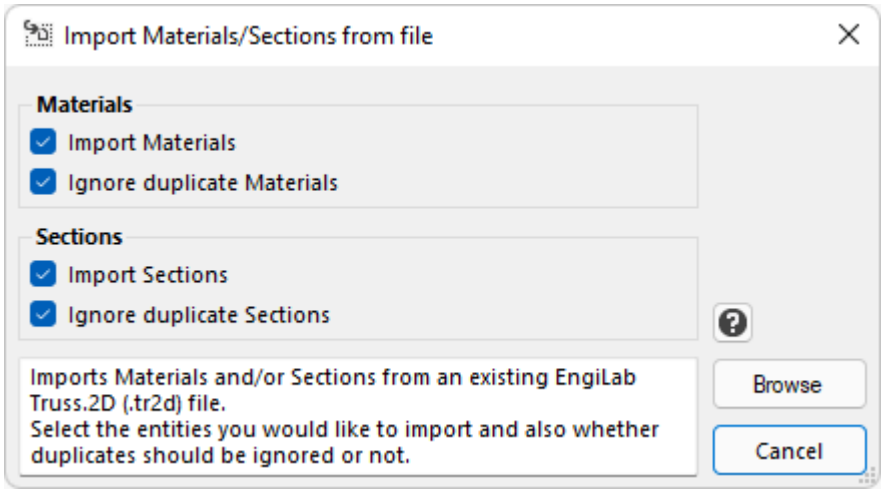
The Tools Menu of the program offers the following options:

- Import Materials/Sections
- Import DXF

3.1 Import Materials/Sections



The user can import **Materials** and/or **Sections** from an existing EngiLab Truss.2D file (**.tr2d** file).

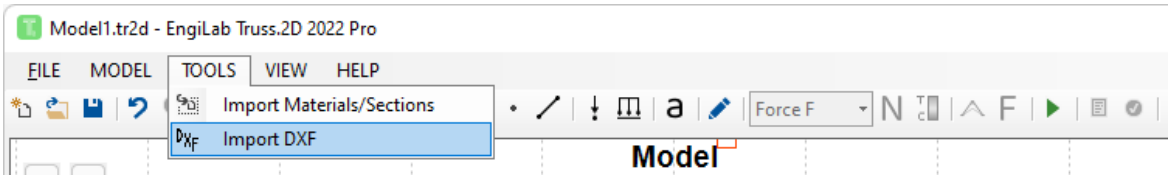


The user can choose to import Materials and/or Sections and also whether duplicate Materials and/or Sections should be ignored.

The Materials and Sections are normally imported with their units. If the unit system used is different between the imported file and the current Model, the program will ask the user whether it should perform unit conversions before importing.

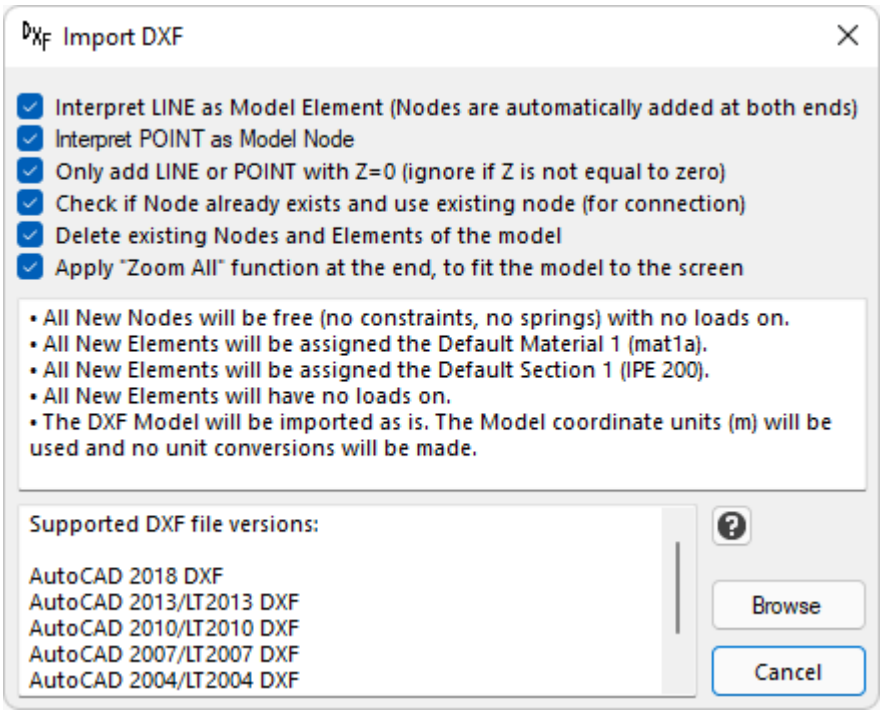
(c) 2022 EngiLab PC

3.2 Import DXF



An alternative way to build a Model quickly, is by importing a DXF file. You can draw your Model in a CAD program using simple LINES for Elements (and optionally POINTS for Nodes) and you can import your drawing into EngiLab Truss.2D as a structural Model.

In order to open the "Import DXF" form, **you first have to define at least one Material and one Section**. All the LINES of the DXF file will be converted to Model Elements assigned the Default Material and the Default Section.



Notes:

- All new Nodes will be free (no constraints, no springs) with no Nodal Loads on them.
- All new Elements will be assigned the Default Material.
- All new Elements will be assigned the Default Section.
- All new Elements will have no Elemental Loads on them.

(c) 2022 EngiLab PC

Supported DXF file versions:

- AutoCAD 2018 DXF
- AutoCAD 2013/LT2013 DXF
- AutoCAD 2010/LT2010 DXF
- AutoCAD 2007/LT2007 DXF
- AutoCAD 2004/LT2004 DXF
- AutoCAD 2000/LT2000 DXF
- AutoCAD R12/LT2 DXF

Most probably the DXF Import feature will work without any problems also with any newer versions of AUTOCAD DXF files.

(c) 2022 EngiLab PC

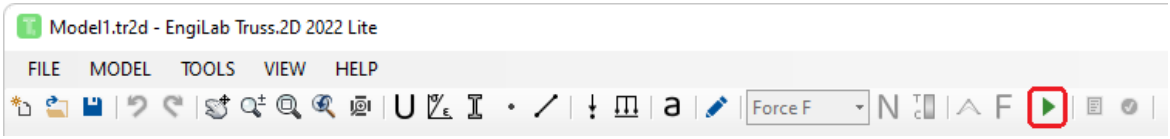
Chapter



Analysis, Results and Validation

4 Analysis, Results and Validation

4.1 Analysis



The analysis is performed using the **Finite Element Method (FEM)** for plane trusses. During the analysis process, the program forms the model effective Stiffness Matrix [K] as well as the vector of the effective External Forces {F}. The supports are also taken into account to form [K] and {F}. Then, the linear equations system {**F**}=[**K**]{**D**} is solved in order for the effective Displacement Vector {**D**} to be calculated:

$\{\mathbf{D}\}=[\mathbf{K}]^{-1}\cdot\{\mathbf{F}\}$

If the model is statically well-defined then the effective Stiffness Matrix [**K**] is reversible (Matrix Determinant greater than zero) and no problems will occur during the analysis process.

Errors during the analysis process

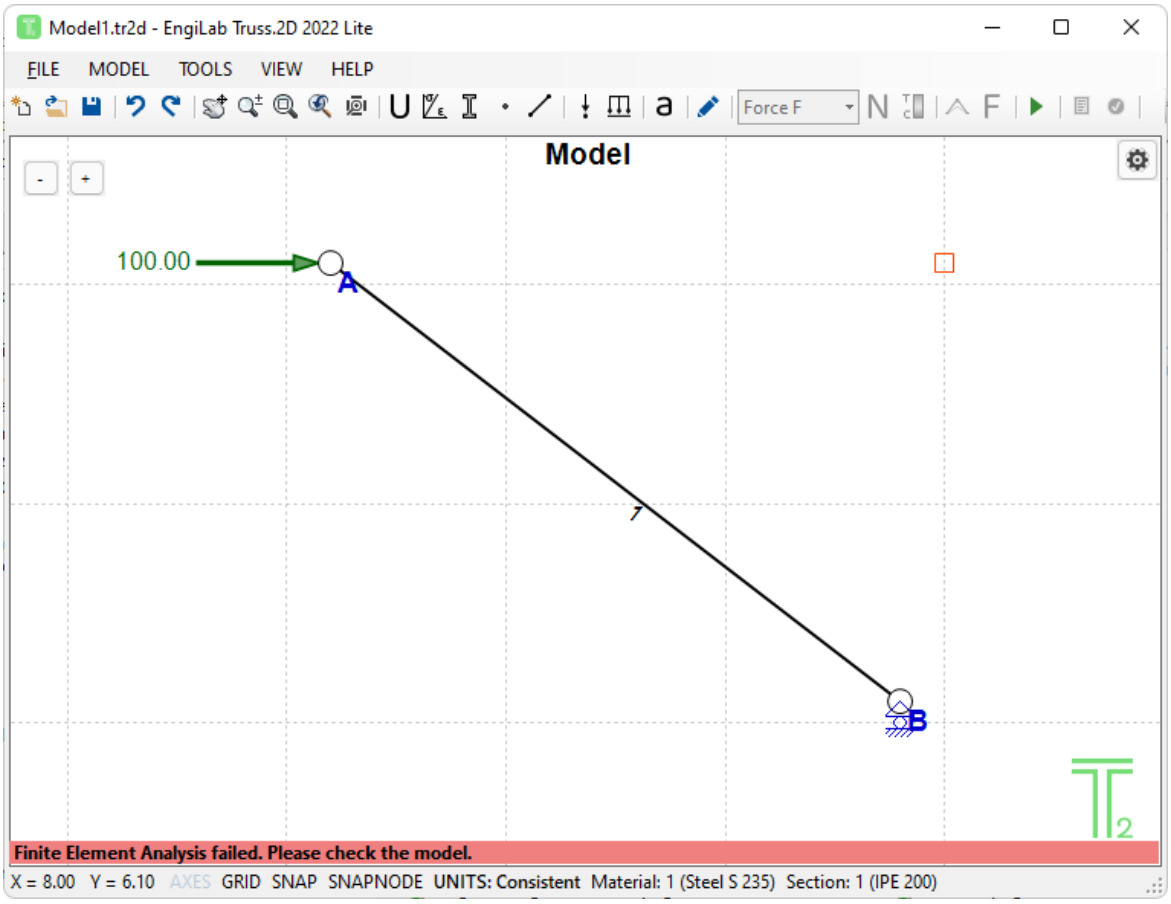
Computational errors may occur during the analysis process. In that case, the program warns the user with the following message:

1. 'Finite Element Analysis failed. Please check the Model.'

This error occurs due to the model stiffness matrix being non-reversible. There may be a zero-element appearing at the matrix diagonal. In this case the determinant of the model effective stiffness matrix is equal to zero, the matrix is non-reversible and the linear system of equations cannot be solved for the displacements. The structure is essentially a mechanism that cannot be analyzed. In this case, check the Model and make sure it is not a mechanism.

(c) 2022 EngiLab PC

Example: An under-constrained Model, such as a member with a roller support, as shown below. The Model can freely rotate and move and equilibrium cannot be reached, by any means. The Model cannot be analyzed and the programs gives us the error message, as shown below.



(c) 2022 EngiLab PC

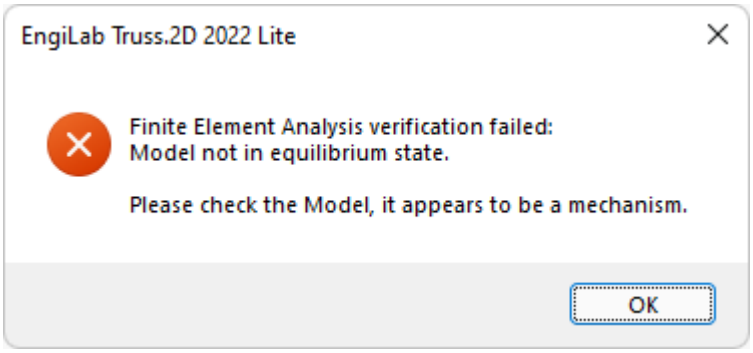
2. 'Finite Element Analysis verification failed: Model not in equilibrium state. Please check the Model, it appears to be a mechanism.'

This case does not have to do with a computational error occurring during the Analysis process. The Analysis has been carried out "successfully" by the computer, and no numerical errors occurred. Yet, after the Analysis is finished, EngiLab Truss.2D performs several verification tests to make sure that the Analysis results are valid and trustworthy. In particular, the program calculates the following, for the entire Model (taking also into account the calculated support reactions):

- Sum of Forces in the X- direction (ΣFX)
- Sum of Forces in the Y- direction (ΣFY)
- Sum of Moments about the (0, 0) point on the plane (ΣM(0,0)).

These calculations take into account the external forces and the calculated (from the Analysis) support reactions. In a well-defined Model that has been successfully analyzed, all three values must be equal to zero, i.e. ΣFX=0, ΣFY=0, ΣM(0,0)=0 so that the Model is in equilibrium state after the external loads have been applied.

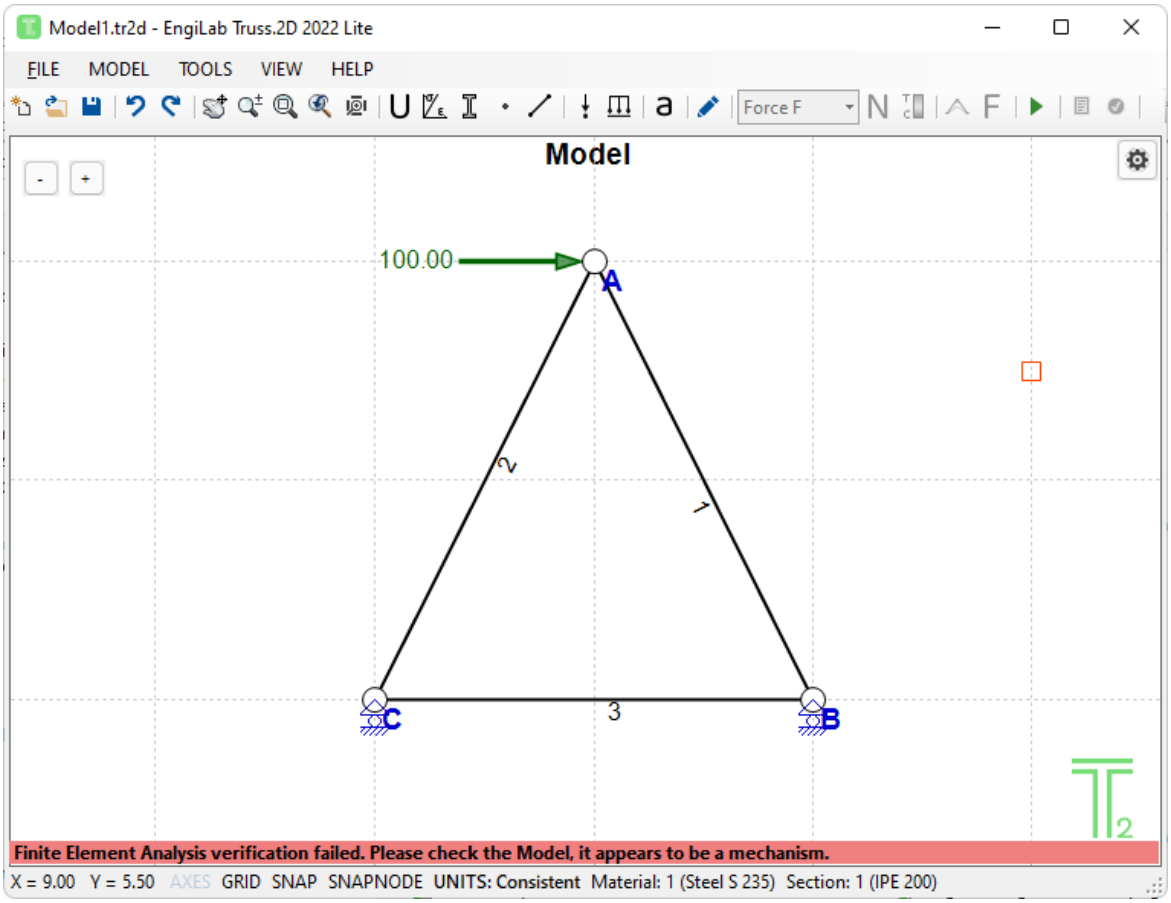
If, after the analysis, any of these values is not equal to zero (given a tolerance value of 10^{-5}, due to rounding errors), then this is a clear indication that something is wrong with the Model and the program will show the following error message.



This means that most probably the structure is a mechanism. In this case the model stiffness matrix is in fact non-reversible but during the analysis process instead of the right zero value, a very small non-zero value appears at the matrix diagonal due to computational rounding errors. As a result, the Analysis solver finds a solution (with no practical interest) to the linear equations system, yet the resultant displacements have excessive values and the Model is NOT in equilibrium state.

This, for example, can happen in the Model shown below. The Model is obviously a mechanism (only two X-Rollers as supports), as was the one above, but this time the computer gives us the second error message, instead of the first.

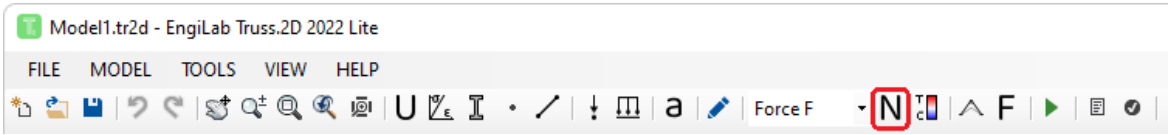
(c) 2022 EngiLab PC



Possible reasons for such error messages:

- There are Nodes that are not connected to the model via Elements.
- The Constraints (supports) are inadequate (under-constrained Model), for example the two models shown above.
- Some parts of the structure are not connected to each other and as a result the forces cannot be transferred from one part of the structure to another.
- Due to some other reason, the structure is a mechanism.

4.2 Axial Diagrams



After setting up the Model and analyzing it (Clicking the 'Analyze' button ▶) you can click N to see the **Axial Diagrams**.

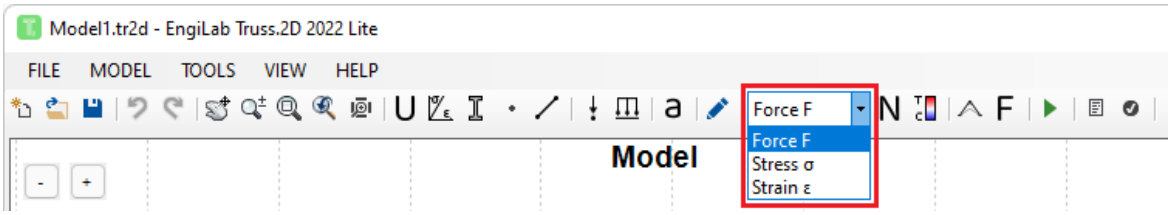
There are 3 types of these Diagrams:

(c) 2022 EngiLab PC

- **Axial Force** Diagram
- **Axial Stress** Diagram
- **Axial Strain** Diagram

Axial Diagrams depict the values of the Forces, Stresses or Strains in a Diagram, for each Element.

You can select one of the Diagrams by clicking on the Dropdown menu to the left of the N button:



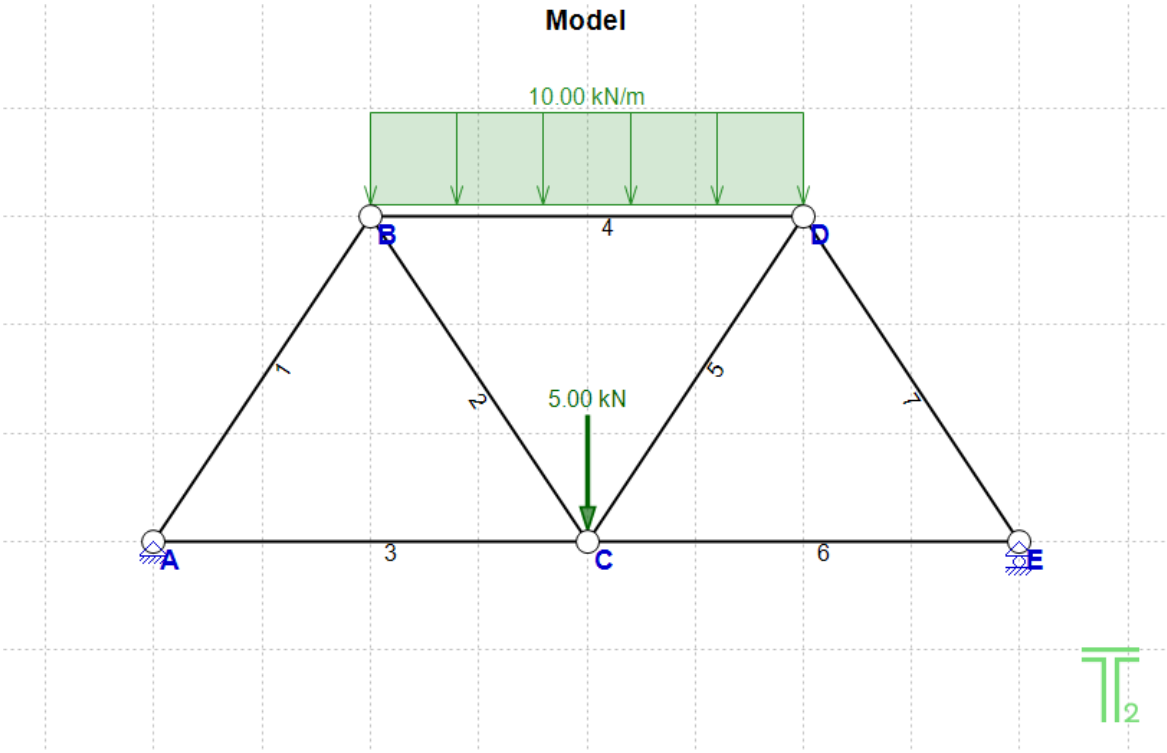
Note: Diagram values are given also on screen, if the mouse pointer hovers over an element. By default, the diagram of a member in **tension** is in **red color** and the diagram of a member in **compression** is in **blue color**. These are the default settings, but the user can change these colors in Colors (Settings). The tone of red/blue is not changed based on the value of the tensile/compressive force/stress/strain, unlike in Color Plots where the color is "proportional" to the value.

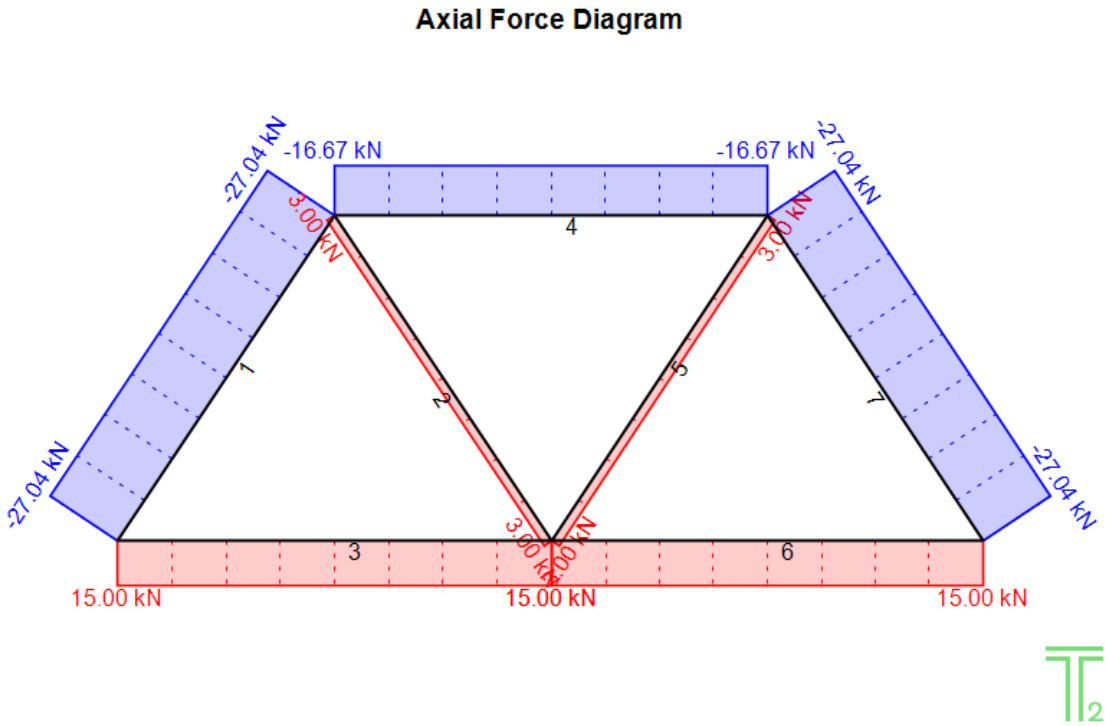
See also:

- Axial Diagrams/Color Plots (Settings)
- Colors (Settings)

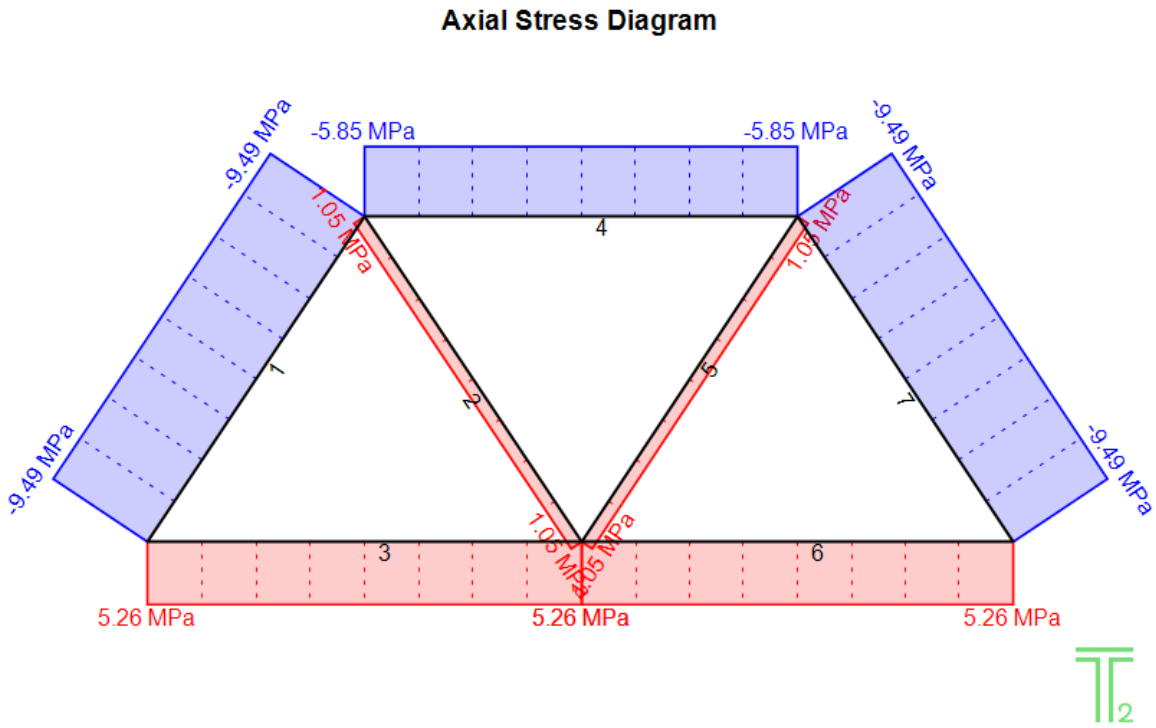
(c) 2022 EngiLab PC

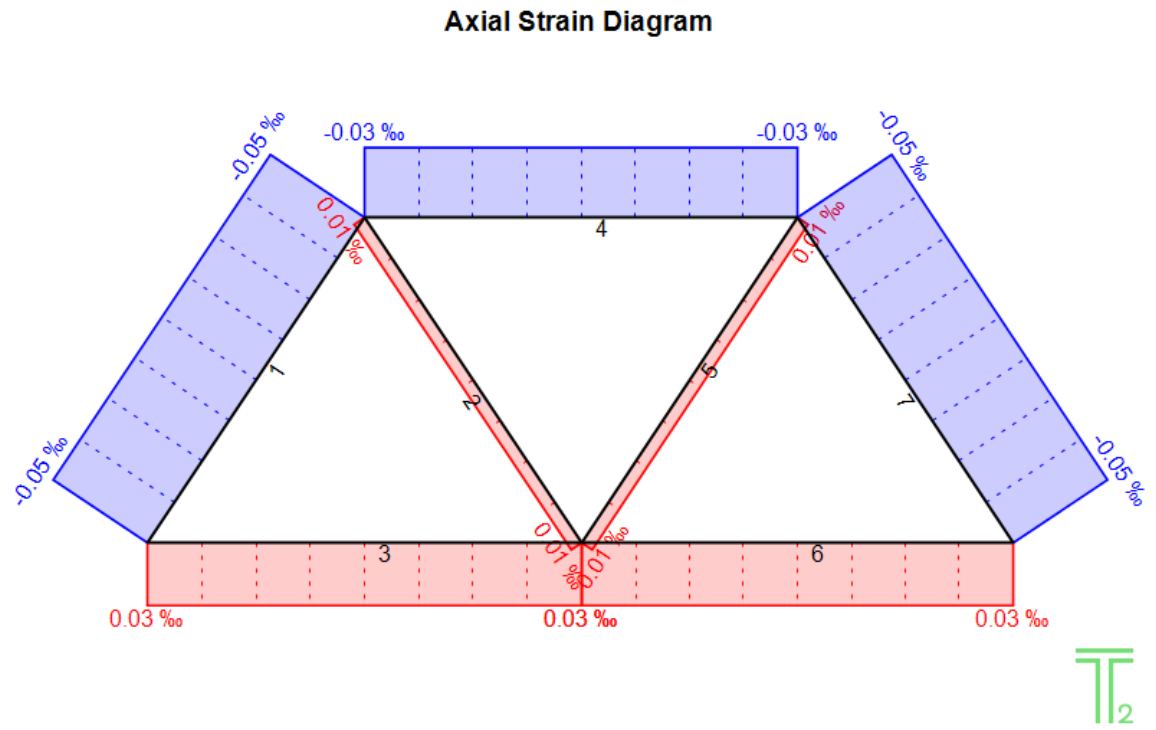
Example



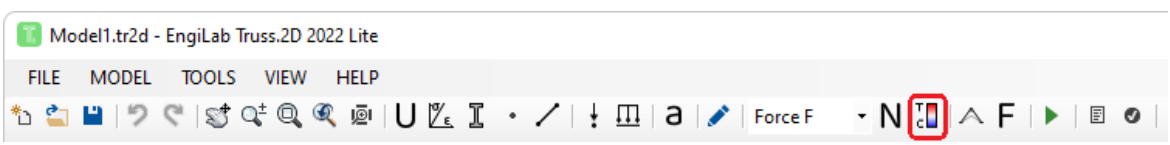


(c) 2022 EngiLab PC





4.3 Color Plots

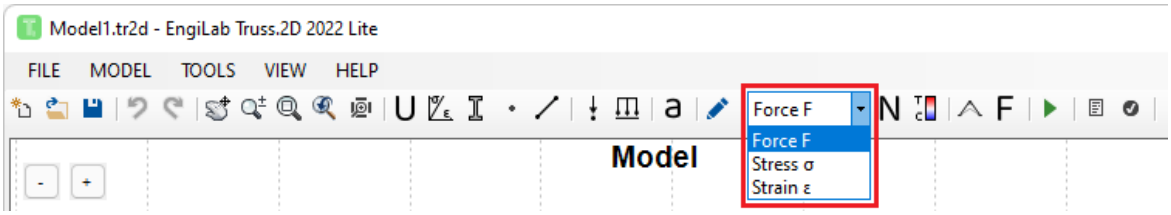


After setting up the Model and analyzing it (Clicking the 'Analyze' button) you can click to see the **Color Plots**.

There are 3 types of these Diagrams:

- **Axial Force** Color Plot
- **Axial Stress** Color Plot
- **Axial Strain** Color Plot

You can select one of the Diagrams by clicking on the Dropdown menu to the left of the N button:



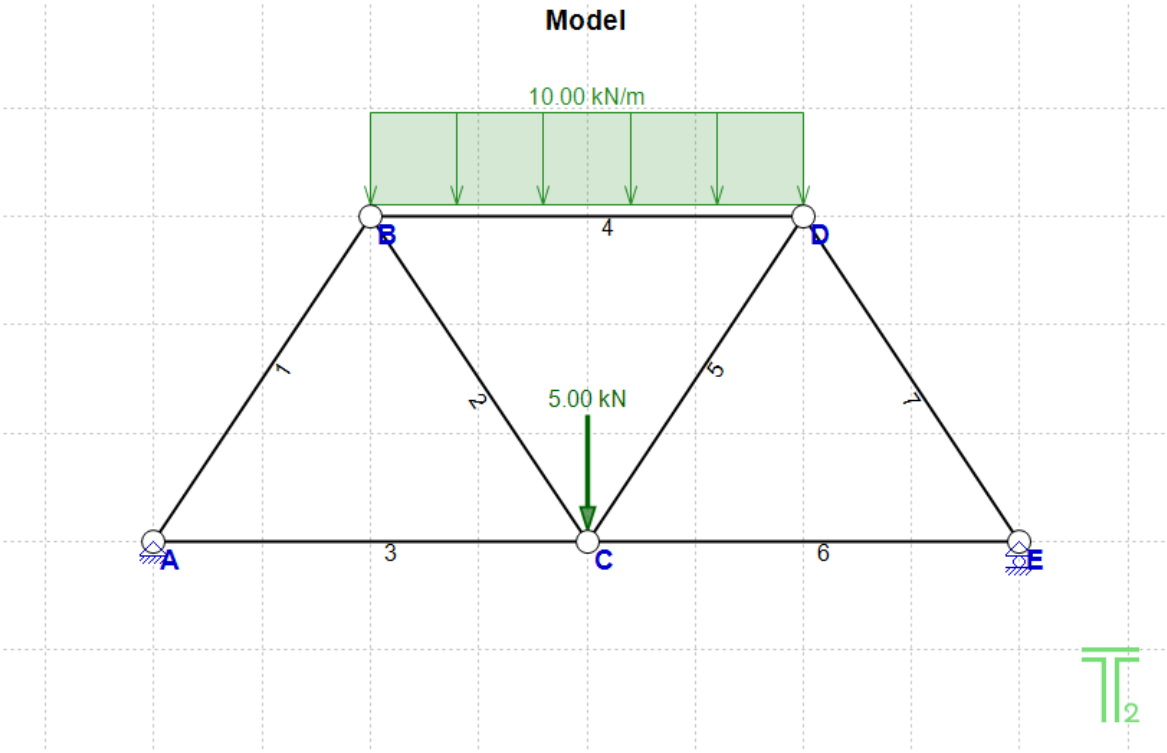
Note: Diagram values are given also on screen, if the mouse pointer hovers over an element. By default, the diagram of a member in **tension** is in **red color** and the diagram of a member in **compression** is in **blue color**. In Color Plots, each member is colored with a color "proportional" to the value of the Force/Stress/Strain. For this, a color for a zero-force member needs also to be defined. By default, zero-force members are plotted in **white color**, matching the background color of the Image. These are the default settings for the three colors, but the user can change these colors in Colors (Settings).

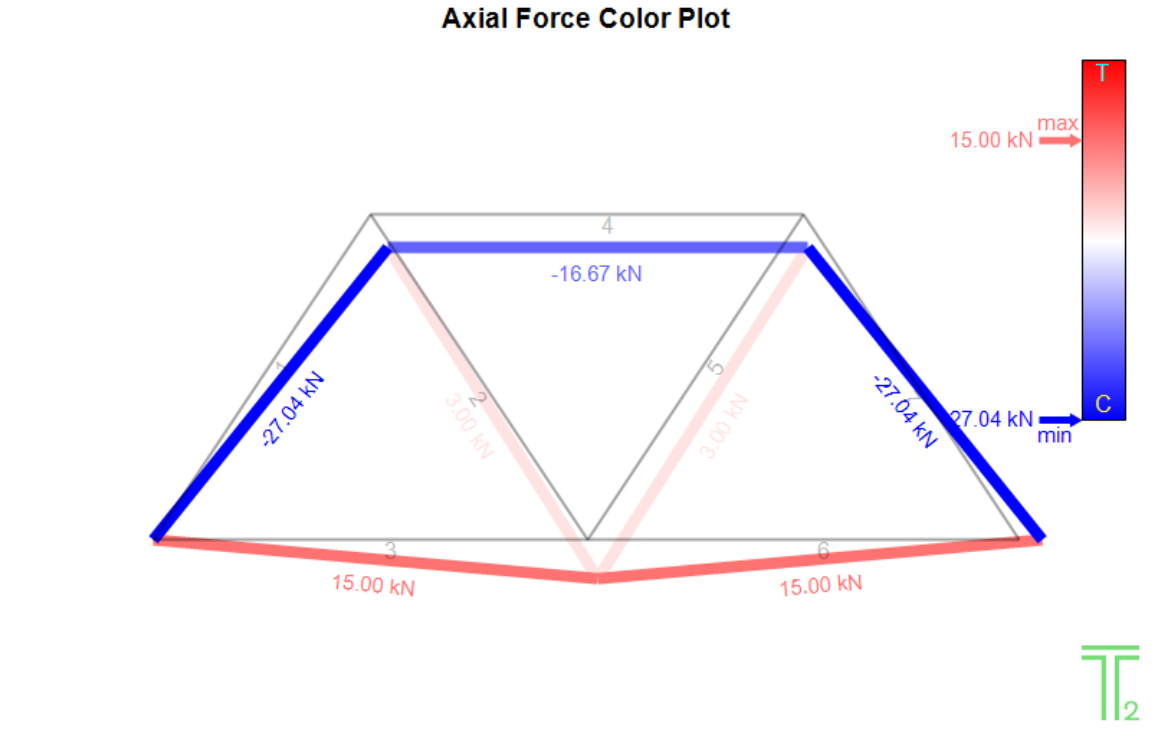
See also:

- Axial Diagrams/Color Plots (Settings)
- Colors (Settings)

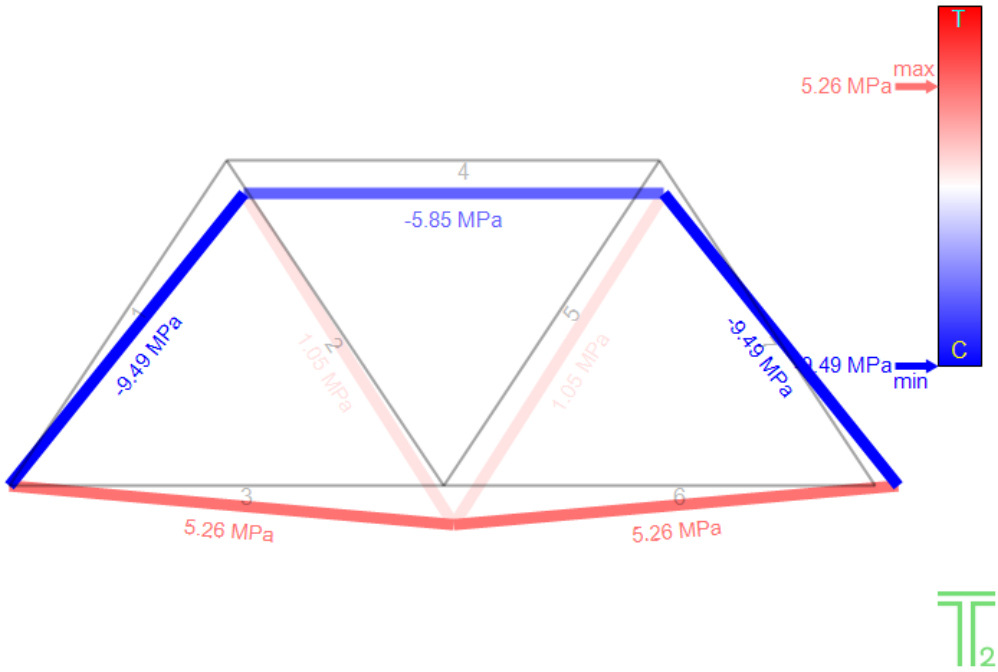
(c) 2022 EngiLab PC

Example

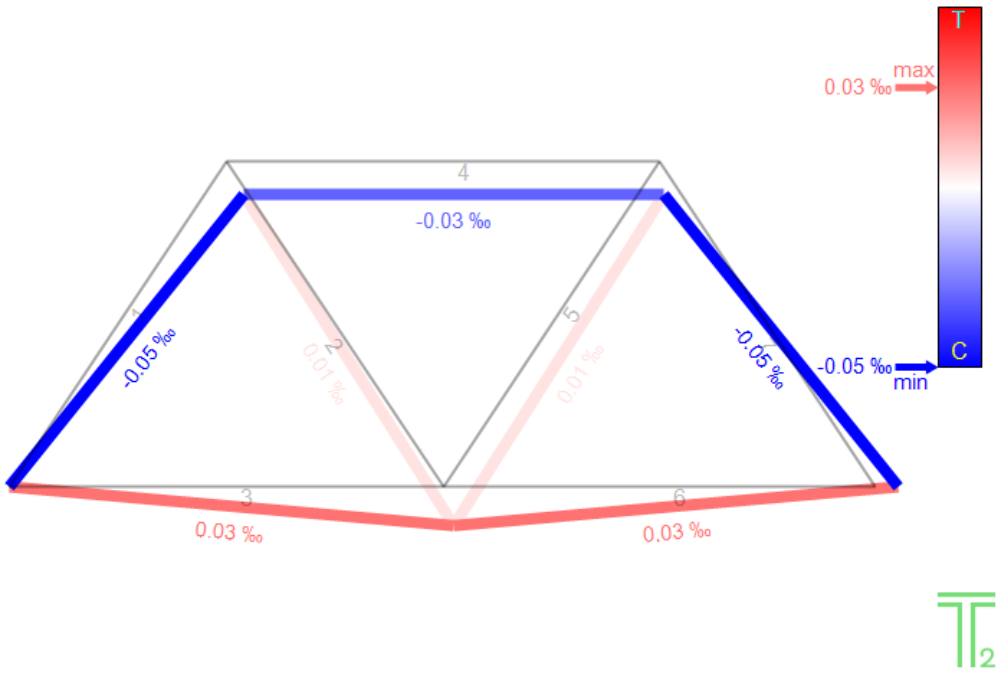

Model
10.00 kN/m
B
D
4
1
2
5
7
5.00 kN
A
3
C
6
E


Axial Force Color Plot
T
15.00 kN
max
4
-16.67 kN
1
-27.04 kN
2
3.00 kN
5
3.00 kN
7
-27.04 kN
27.04 kN
min
C
3
15.00 kN
6
15.00 kN

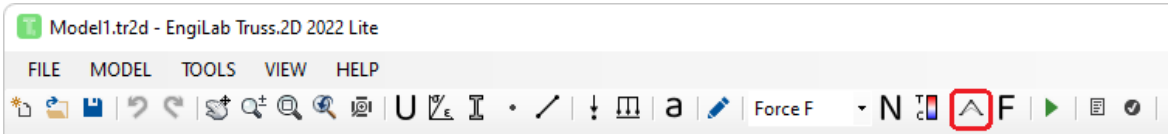
Axial Stress Color Plot



Axial Strain Color Plot



4.4 Deformation



(c) 2022 EngiLab PC

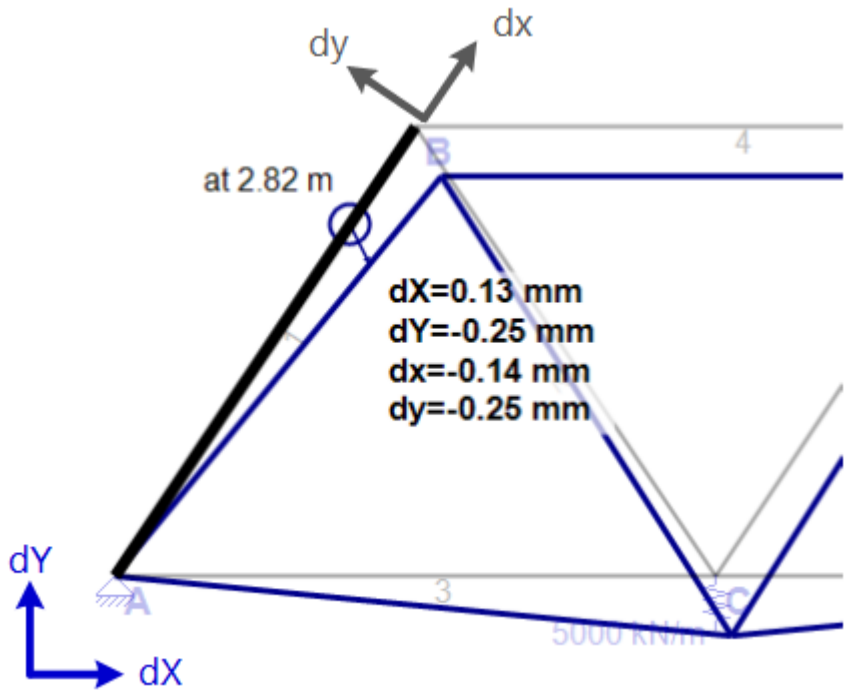
After setting up the model and analyzing it (Clicking the 'Analyze' button ▶) you can click ⩓ to see the deformed shape of the Model.

- ⩓: Deformation

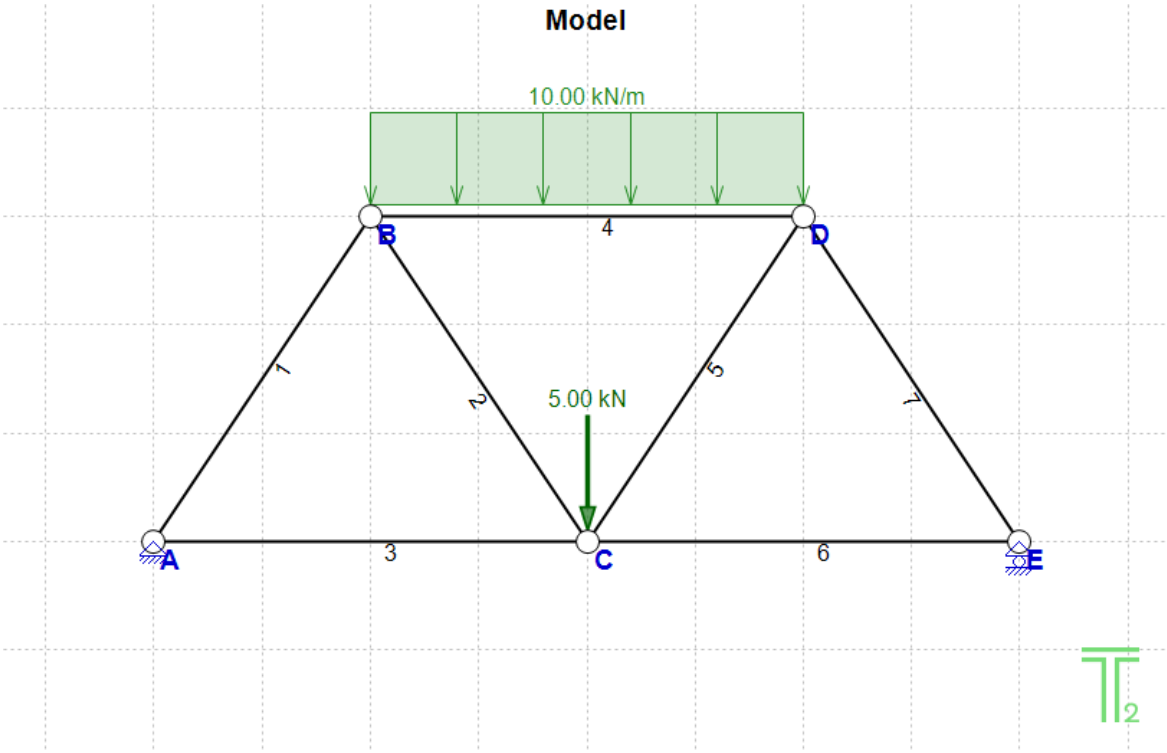
Note: Deformation values are given also on screen, if the mouse pointer hovers over an element. The values that are given on screen are the **dX**, **dY**, **dx** and **dy** displacements of the corresponding point of each element.

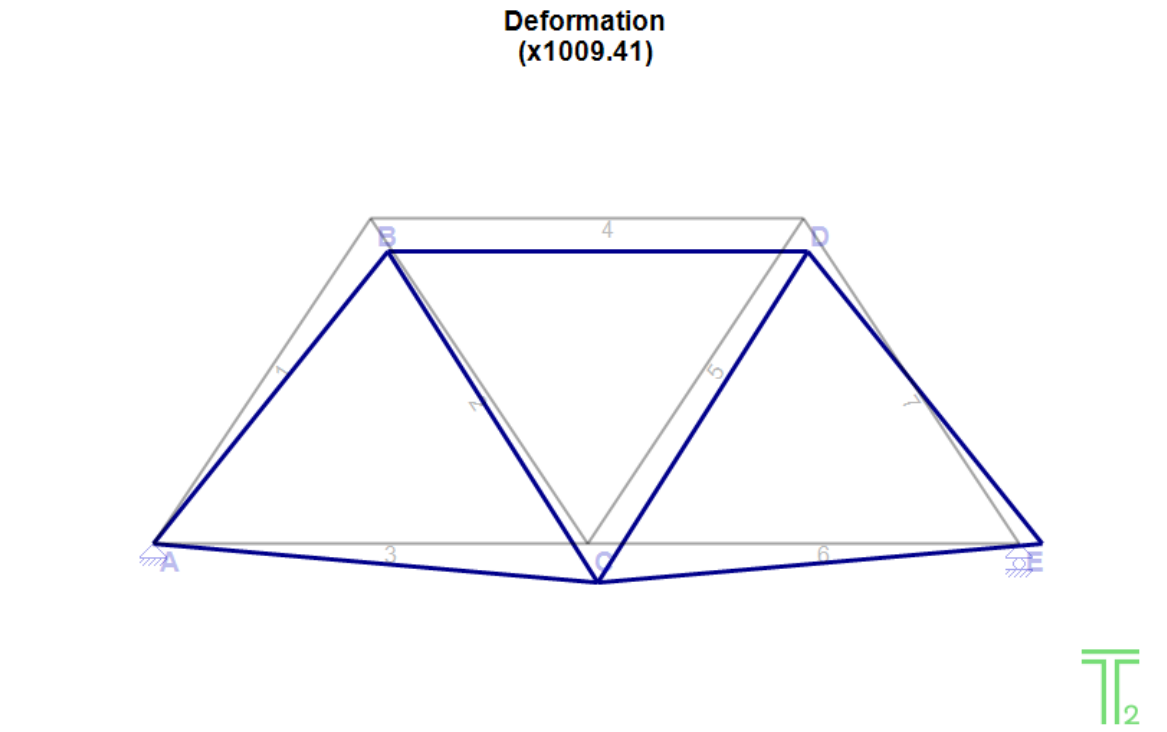
- **dX** and **dY** are the X- and Y- displacements of a point, respectively, in **Global axes**. For example a positive dX means displacement to the right, while a positive dY means displacement upwards.
- **dx** and **dy** are the x- and y- displacements of a point, respectively, in **Local Element Axes**, where x is defined from the Start Node i and the End Node j of the Element and y is perpendicular to it.

See the illustrative example below which explains this convention.



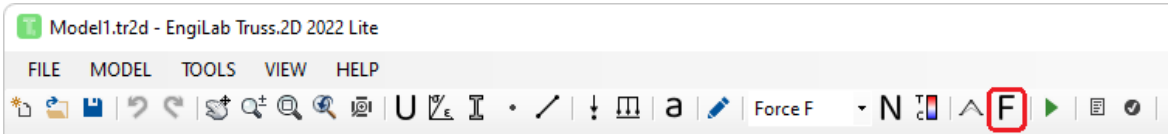
Example





(c) 2022 EngiLab PC

4.5 Free Body Diagram



After setting up the model and analyzing it (Clicking the 'Analyze' button ▶) you can click F to see the **Free Body Diagram** of the Model.

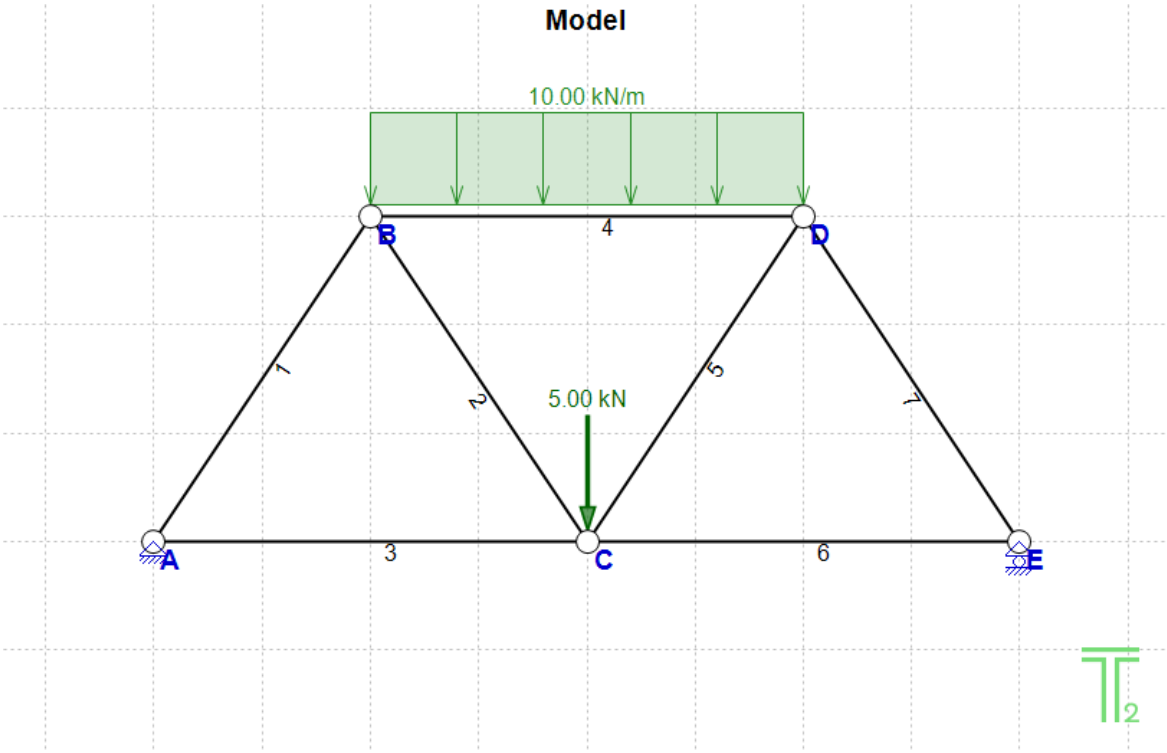
- F: Free Body Diagram

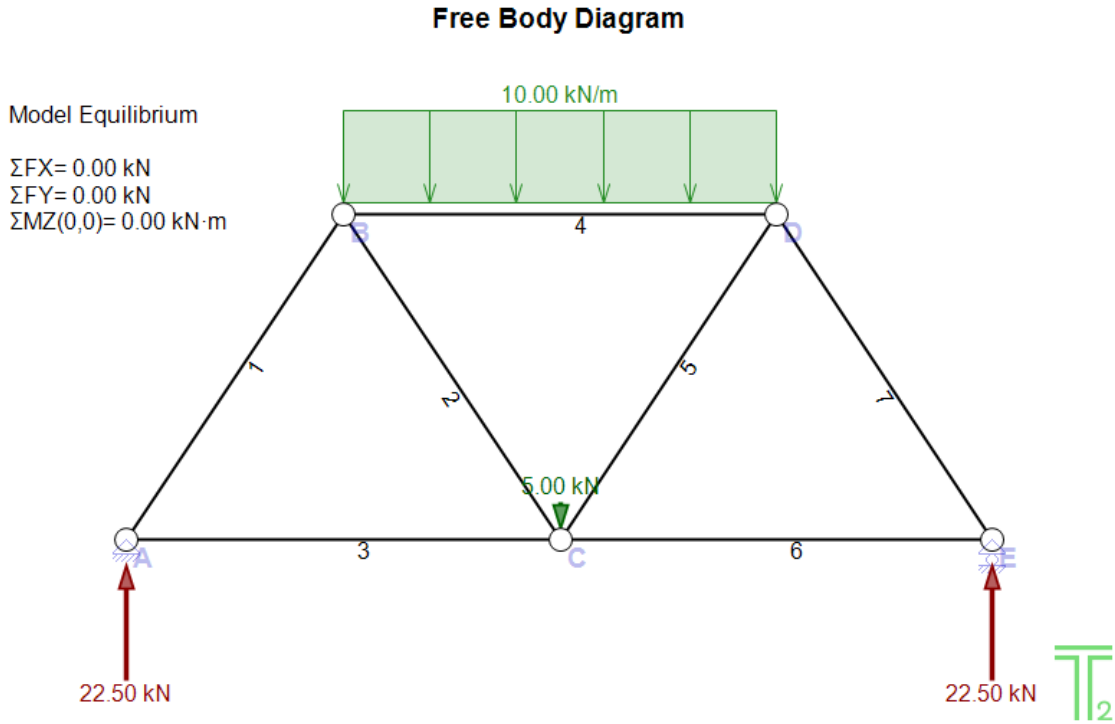
The Free Body Diagram shows the Model, the external loading, the support reactions and also the calculations of the equilibrium of the Model.

Note: If the mouse pointer hovers over an element, the program shows the corresponding Axial Force at the corresponding point along the element. The program shows also the Element End Forces of the specific Element.

(c) 2022 EngiLab PC

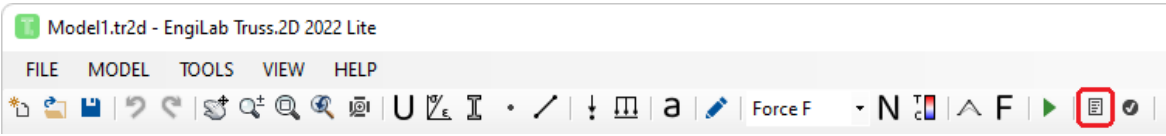
Example





(c) 2022 EngiLab PC

4.6 Analysis Results Form



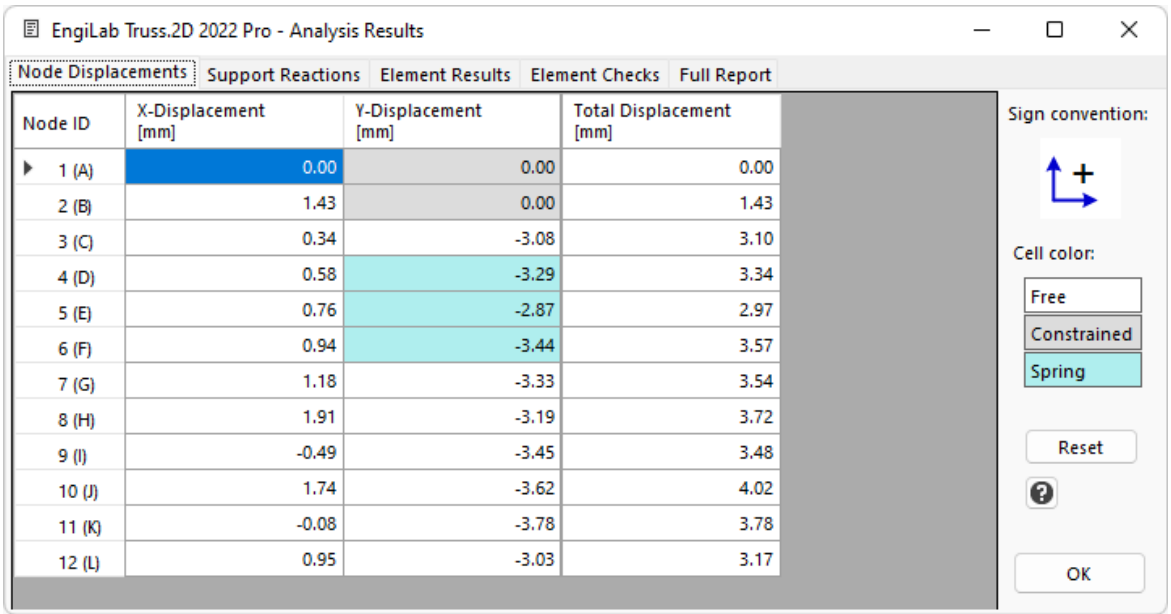
By clicking the Analysis Results button, you get a new form where you can see the following:

- Node Displacements
- Support Reactions
- Element Results
- Element Checks
- Full Report

4.6.1 Node Displacements

This tab shows the Node Displacements of the Model. The sign convention is according to the Global Axes, as shown in the figure.

- Constrained DOFs (supported) and Spring DOFs are marked with special colors.



Note:

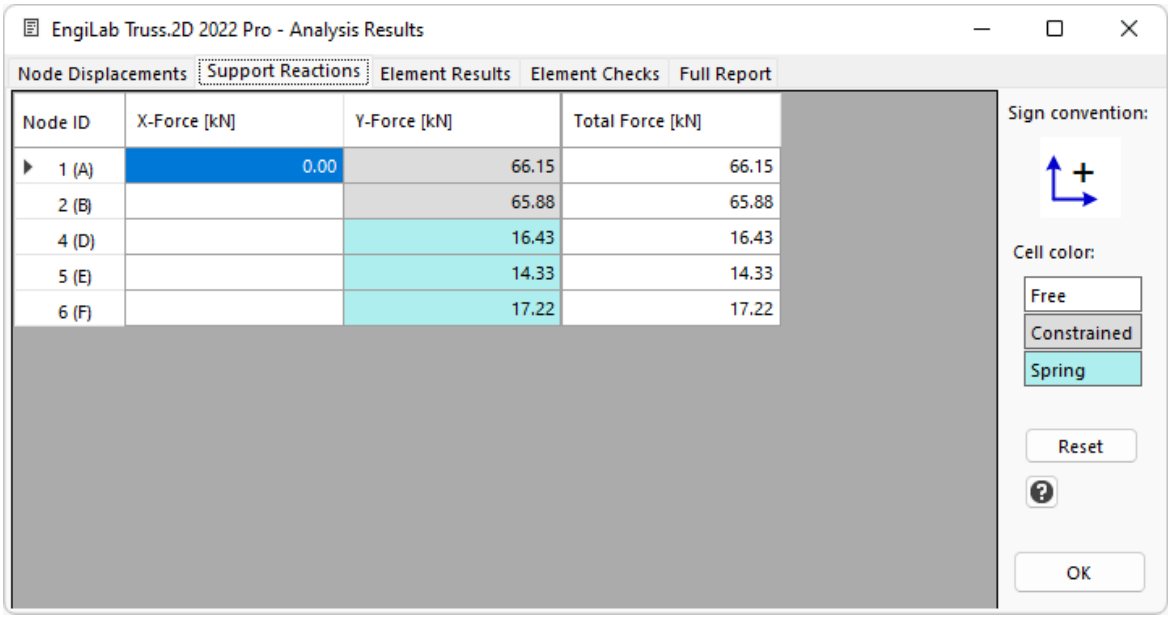
- All three columns (X-Displacement, Y-Displacement and Total Displacement) are **sortable columns**.

- By clicking on the header row, the user can sort the results (Nodes numbering and the corresponding displacements) based on one of the columns. This is useful to find, for example, the max. or min. X-, Y- or Total Displacement.
- After sorting, to get back the original numbering of the Nodes, simply click the "**Reset**" button.

4.6.2 Support Reactions

This tab shows the Support Reactions (Constraints or Springs) of the Model. The sign convention is according to the Global Axes, as shown in the figure.

- Constrained DOFs (supported) and Spring DOFs are marked with special colors.



Note:

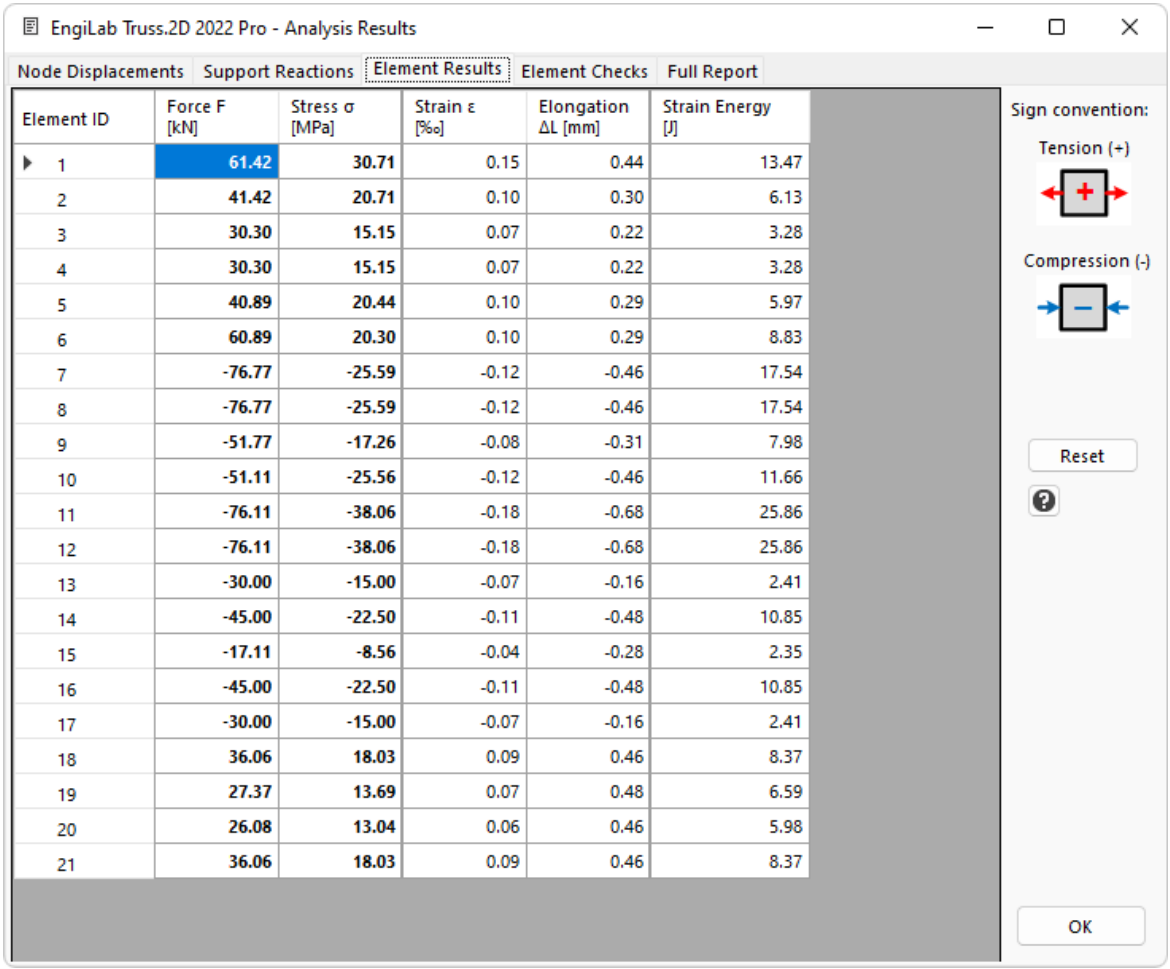
- All three columns (X-Force, Y-Force and Total Force) are **sortable columns**.
- By clicking on the header row, the user can sort the results (Nodes numbering and the corresponding support forces) based on one of the columns. This is useful to find, for example, the max. or min. X-, Y- or Total Force.
- After sorting, to get back the original numbering of the Nodes, simply click the "**Reset**" button.

(c) 2022 EngiLab PC

4.6.3 Element Results

This tab shows the Element Results. The following quantities are reported for each Element:

1. The **Axial Force, F**.
2. The **Axial Stress, σ=F/A**, where A is the Section Area.
3. The **Axial Strain, ε=σ/E**, where E is the Elasticity Modulus of the Material.
4. The **Elongation, ΔL=L*σ/E**, where L is the original Length of the Element (i.e. in the undeformed state).
5. The **Strain Energy =1/2*(E*A/L)*ΔL² = F²*L/(2*E*A)**, which is the energy stored in the Element in the deformed state.



Element ID	Force F [kN]	Stress σ [MPa]	Strain ε [‰]	Elongation ΔL [mm]	Strain Energy [J]
1	61.42	30.71	0.15	0.44	13.47
2	41.42	20.71	0.10	0.30	6.13
3	30.30	15.15	0.07	0.22	3.28
4	30.30	15.15	0.07	0.22	3.28
5	40.89	20.44	0.10	0.29	5.97
6	60.89	20.30	0.10	0.29	8.83
7	-76.77	-25.59	-0.12	-0.46	17.54
8	-76.77	-25.59	-0.12	-0.46	17.54
9	-51.77	-17.26	-0.08	-0.31	7.98
10	-51.11	-25.56	-0.12	-0.46	11.66
11	-76.11	-38.06	-0.18	-0.68	25.86
12	-76.11	-38.06	-0.18	-0.68	25.86
13	-30.00	-15.00	-0.07	-0.16	2.41
14	-45.00	-22.50	-0.11	-0.48	10.85
15	-17.11	-8.56	-0.04	-0.28	2.35
16	-45.00	-22.50	-0.11	-0.48	10.85
17	-30.00	-15.00	-0.07	-0.16	2.41
18	36.06	18.03	0.09	0.46	8.37
19	27.37	13.69	0.07	0.48	6.59
20	26.08	13.04	0.06	0.46	5.98
21	36.06	18.03	0.09	0.46	8.37

Sign convention:

In EngiLab Truss.2D, as shown in the Figure:

- **Tensile** Element Forces, Stresses and Strains are reported as **Positive**.
- **Compressive** Element Forces, Stresses and Strains are reported as **Negative**.

Note:

- All columns are **sortable columns**.
- By clicking on the header row, the user can sort the results based on one of the columns. This is useful to find, for example, the max. or min. Force, Stress, etc.
- After sorting, to get back the original numbering of the Elements, simply click the "**Reset**" button.

(c) 2022 EngiLab PC

4.6.4 Element Checks

This tab shows the Element Checks. The following checks are made by the program and reported for each Element:

1. The **Axial Force, F**.

2. The **Euler's Critical Buckling Load, BL = $-\pi^2*E*I/L^2$**.

BL is always reported with a negative sign (compression), as it is the compressive load at which a slender member will suddenly buckle. Members under tension never buckle.

The above formula is valid for truss members, as truss members are pinned at both ends (free rotation at both ends).

I is the Moment of Inertia of the Section which is given by the user (in Sections) and is only used for this check (it is not used for the Analysis and it does not affect it).

I is an optional property in Sections. If I is not given by the user for an Element, then the Buckling check is not performed and "**N/A**" will be reported for both the Critical Buckling Load and the corresponding Element check.

3. The **Utilization Ratio for Buckling, F/BL**, which is reported only for Elements under compression.

If F/BL ≥ 1 (or ≥100%) for a compressive member, the member fails under buckling and it is reported with red color in the table.

4. The **Axial Stress, σ=F/A**, where A is the Section Area.

5. The **Yield Stress, fy**, for the Element Material, which is supposed to be the same (in absolute terms) for tension and compression.

fy is given by the user (in Materials) and is only used for this check (it is not used for the Analysis and it does not affect it).

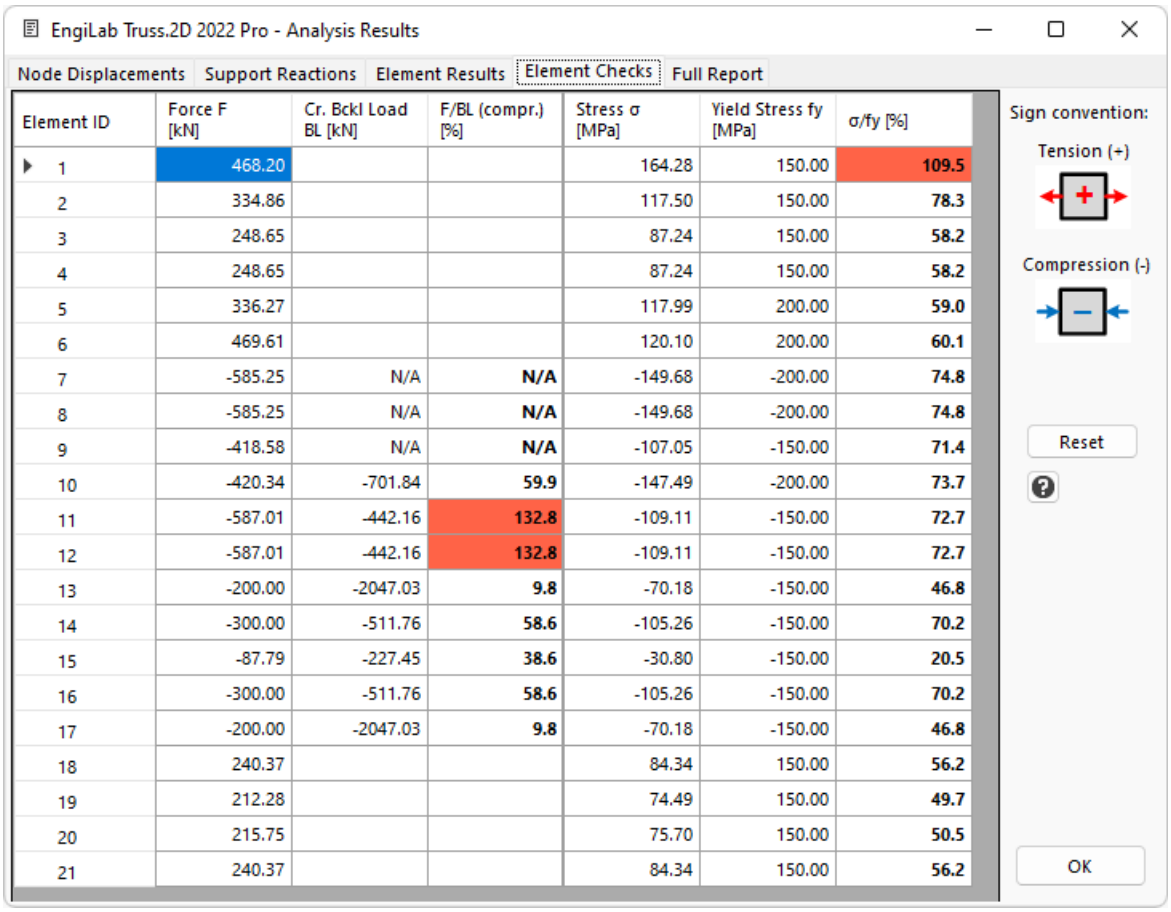
fy is an optional property in Materials. If fy is not given by the user for an Element, then the Yield check is not performed and "**N/A**" will be reported for both the Yield Stress and the corresponding Element check.

If F≥0 (Element in tension or with zero force), then fy is reported with a positive sign. If F<0 (member in compression) then fy for the particular Element is reported with a negative sign, to match the force of the Element.

6. The **Utilization Ratio for Stress, σ/fy**, which is reported for all Elements (under tension or compression).

(c) 2022 EngiLab PC

If σ/fy ≥ 1 (or ≥100%), the member yields, i.e. it fails under stress as its stress is higher than the yield stress, and it is reported with red color in the table.



Element ID	Force F [kN]	Cr. Bckl Load BL [kN]	F/BL (compr.) [%]	Stress σ [MPa]	Yield Stress fy [MPa]	σ/fy [%]
1	468.20			164.28	150.00	**109.5**
2	334.86			117.50	150.00	**78.3**
3	248.65			87.24	150.00	**58.2**
4	248.65			87.24	150.00	**58.2**
5	336.27			117.99	200.00	**59.0**
6	469.61			120.10	200.00	**60.1**
7	-585.25	N/A	**N/A**	-149.68	-200.00	**74.8**
8	-585.25	N/A	**N/A**	-149.68	-200.00	**74.8**
9	-418.58	N/A	**N/A**	-107.05	-150.00	**71.4**
10	-420.34	-701.84	**59.9**	-147.49	-200.00	**73.7**
11	-587.01	-442.16	**132.8**	-109.11	-150.00	**72.7**
12	-587.01	-442.16	**132.8**	-109.11	-150.00	**72.7**
13	-200.00	-2047.03	**9.8**	-70.18	-150.00	**46.8**
14	-300.00	-511.76	**58.6**	-105.26	-150.00	**70.2**
15	-87.79	-227.45	**38.6**	-30.80	-150.00	**20.5**
16	-300.00	-511.76	**58.6**	-105.26	-150.00	**70.2**
17	-200.00	-2047.03	**9.8**	-70.18	-150.00	**46.8**
18	240.37			84.34	150.00	**56.2**
19	212.28			74.49	150.00	**49.7**
20	215.75			75.70	150.00	**50.5**
21	240.37			84.34	150.00	**56.2**

Sign convention:

In EngiLab Truss.2D, as shown in the Figure:

- **Tensile** Element Forces, Stresses and Strains are reported as **Positive**.
- **Compressive** Element Forces, Stresses and Strains are reported as **Negative**.

Note:

- All columns are **sortable columns**.
- By clicking on the header row, the user can sort the results based on one of the columns. This is useful to find, for example, the max. or min. Utilization Ratio, etc.
- After sorting, to get back the original numbering of the Elements, simply click the "**Reset**" button.

(c) 2022 EngiLab PC

4.6.5 Full Report

This tab shows all the Analysis results in Rich Text Format (RTF).

- The Document is editable. The user can edit the text and can even resize the images, if needed.
- The user can easily copy the text and/or images of the document and paste it anywhere (for example in MS Word) for further processing and printing.
- Clicking the "**Save**" button will save the document in **RTF format**. The file can be opened with any text editor, such as MS Word, for **further processing and printing**.
- On the right, there are settings where the user can select what to include in the report.
- The user can choose the (max) picture size (in Pixels) for the included images.
- The Analysis Validation is not included in this Report.

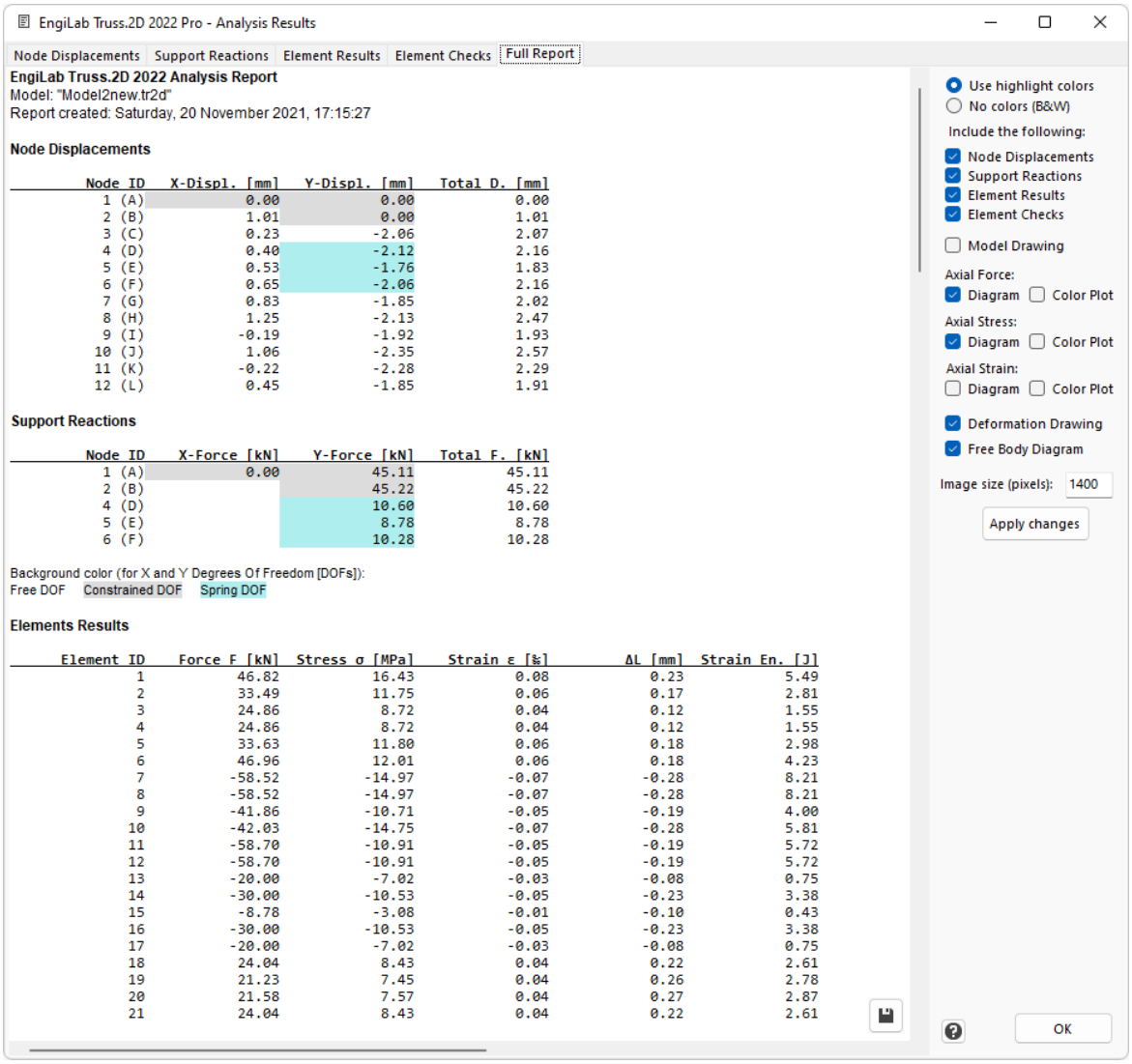
The following **4 tables** can be included in the Report:

- Node Displacements
- Support Reactions
- Element Results
- Element Checks

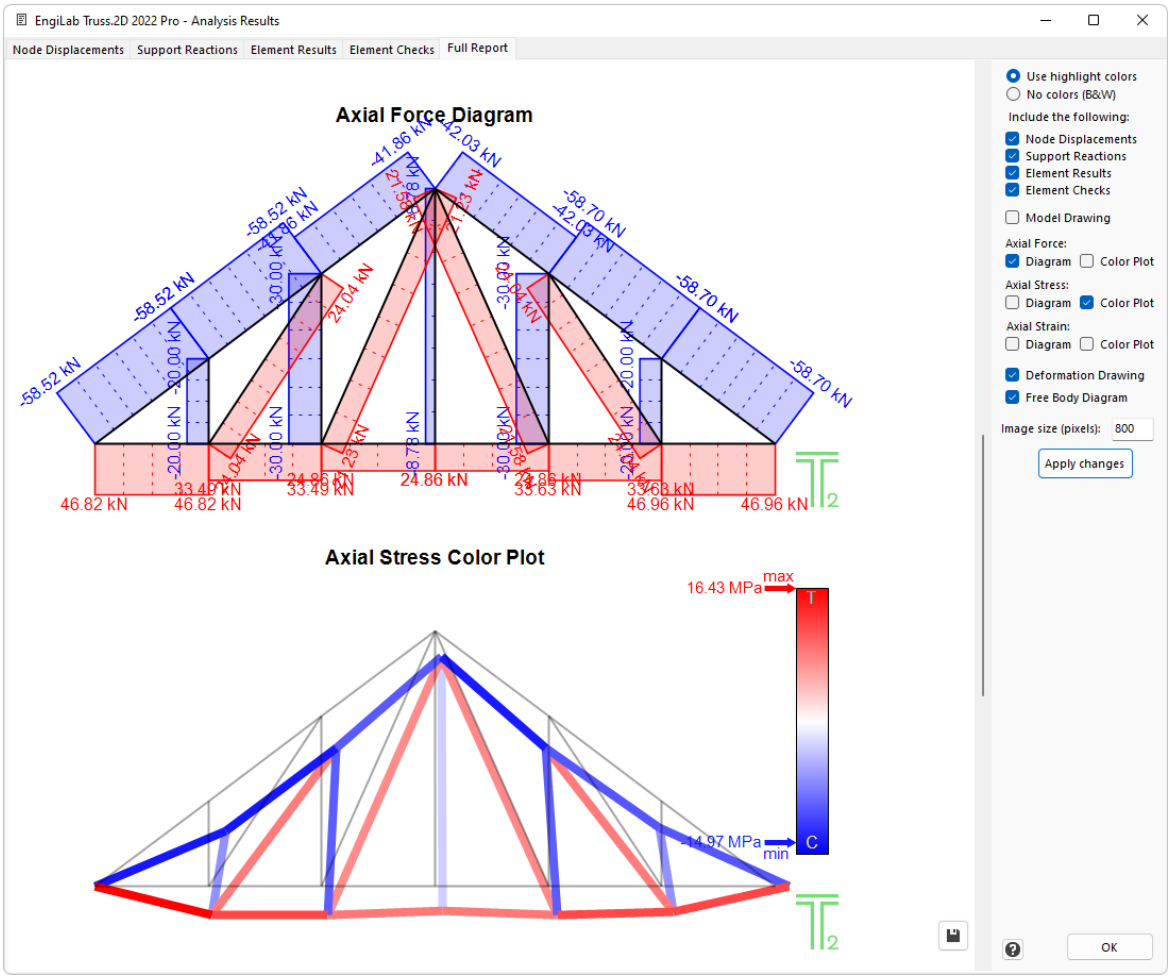
The following **9 images** can be included in the Report:

- Model Drawing
- Axial Force Diagram, Axial Stress Diagram, Axial Strain Diagram
- Axial Force Color Plot, Axial Stress Color Plot, Axial Strain Color Plot
- Deformation Drawing
- Free Body Diagram

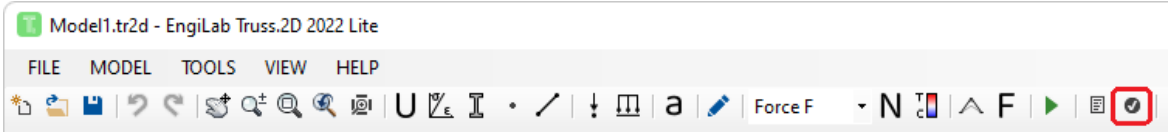
(c) 2022 EngiLab PC



(c) 2022 EngiLab PC



4.7 Analysis Validation Form



By clicking the Analysis Validation button , you get a new form where you can see the Analysis Validation results.

If the Analysis results are correct, then the Model and all its parts (joints and members) should be in equilibrium, while energy should be conserved, according to the principle of energy conservation.

The following tabs are available in the form:

(c) 2022 EngiLab PC

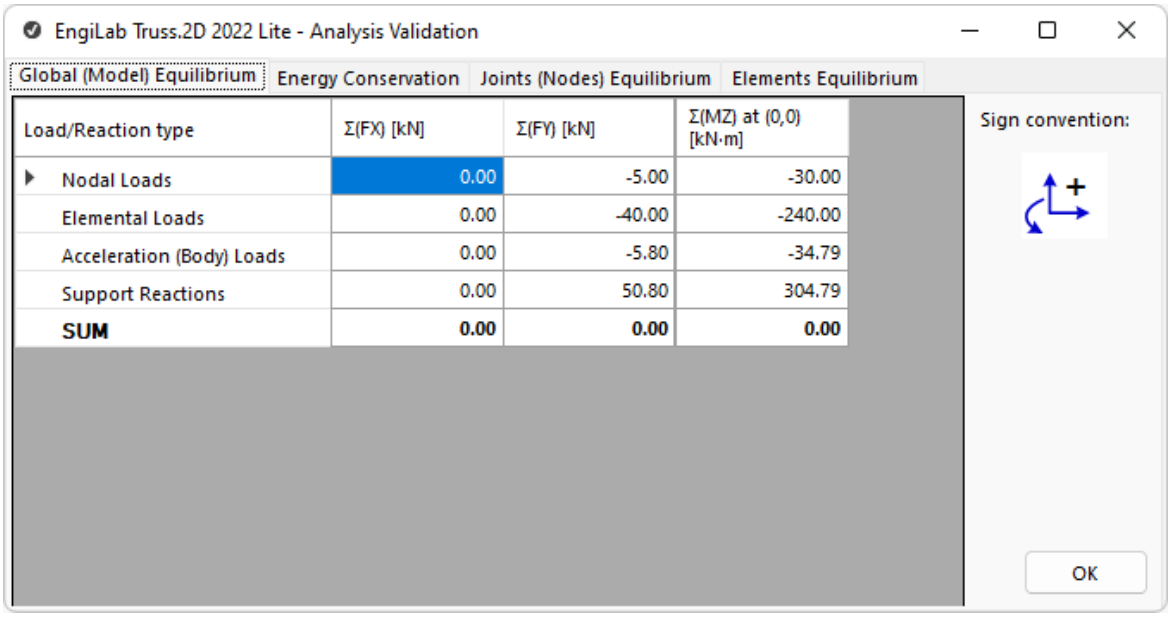
- Global (Model) Equilibrium
- Energy Conservation
- Joints (Nodes) Equilibrium
- Elements Equilibrium

4.7.1 Global (Model) Equilibrium

The program calculates the **Sum of X-Forces Σ(FX)**, the **Sum of Y-Forces Σ(FY)** and the **Sum of Bending Moments Σ(MZ)** (at Point X=0, Y=0) for the whole Model, for the Nodal Loads, Elemental Loads, Acceleration (Body) Loads and the Support Reactions. All three Sums (see the last row of the table, in bold) have to be zero if the Model is in Equilibrium.

Important: If you see values other than zero in the last row (SUM, in bold), then there is a problem with the Model and the Model needs to be checked. The Model is probably a mechanism and any results should not be taken into account. This is already checked by the program, every time an Analysis is conducted, so theoretically it will never happen. But the calculations are there for you to check and verify.

Note: The sign convention is according to the Global Axes, as shown in the figure.



(c) 2022 EngiLab PC

4.7.2 Energy Conservation

This tab shows the calculations related to **Energy Conservation**. The law of conservation of energy states that the total energy of an isolated system remains constant; it is said to be conserved over time. This is the case in a 2D Truss system. This is an important concept from the science of kinetics which simply states that the work done on a body is equal to the energy expended on it.

The program calculates the **work of all external forces**, in both X- and Y- directions:

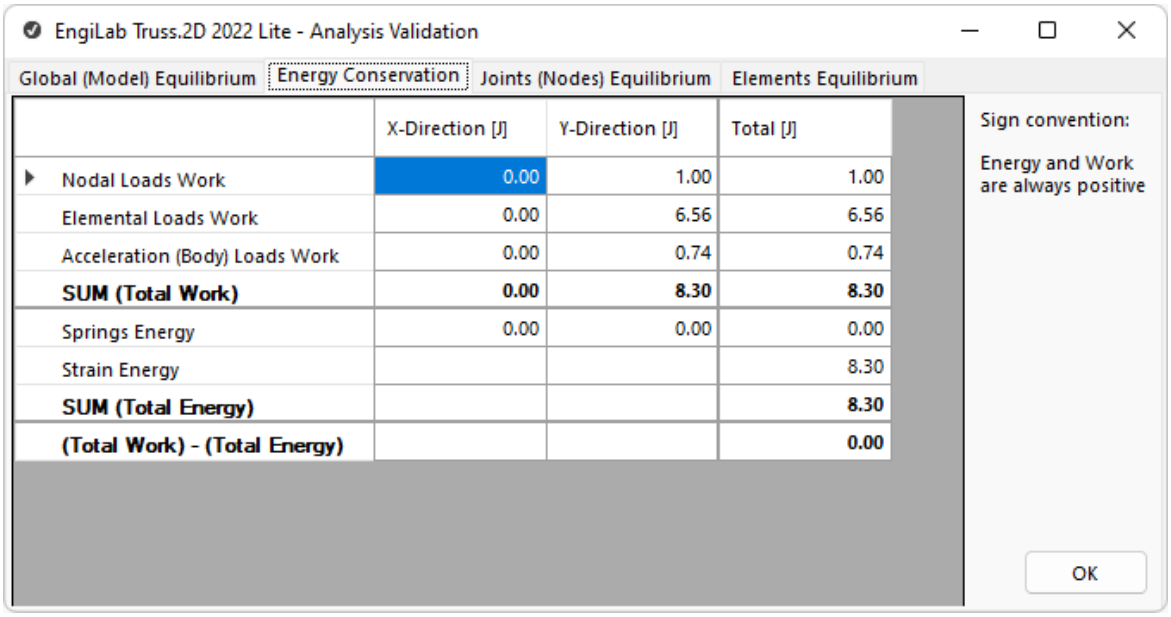
- **Work of Nodal Loads**
- **Work of Elemental Loads**
- **Work of Acceleration (Body) Loads**

Elemental Loads and Acceleration (Body) Loads are handled as equivalent Nodal Loads for the Truss structure, as already mentioned in section What is a 2D Truss?

The program also calculates the Energy stored in the system, i.e. the following quantities:

- **Springs Energy**. It is the energy stored in springs due to Model deformation
- **Strain Energy** of all members. It is the energy stored in every Truss member due to its axial deformation.

If the analysis is successful and the model is in equilibrium, the work offered to the System by the external forces needs to be exactly the same as the Energy stored in the System as Spring Energy and Elastic Strain Energy. The program calculates the Total Work and the Total stored Energy and subtracts them. The result has to be zero or else there is a problem with the Model.



The formulas used for the calculation of the internal energy stored in the System are the following:

Spring Energy $=1/2*K*x^2$,

where K is the stiffness of the spring (spring constant) and x is its extension or compression, i.e. the displacement of the corresponding DOF

Strain Energy $=1/2*(E*A/L)*\Delta L^2 = F^2*L/(2*E*A)$,

where E is the Elasticity Modulus of the Material, A is the Section Area, L is the original Length of the Element (i.e. in the undeformed state), ΔL is the Elongation of the Element, and F is its Axial Force.

The work of Nodal Loads (and the one of the equivalent Nodal Loads due to Elemental Loads and Body Loads) is calculated with the formula:

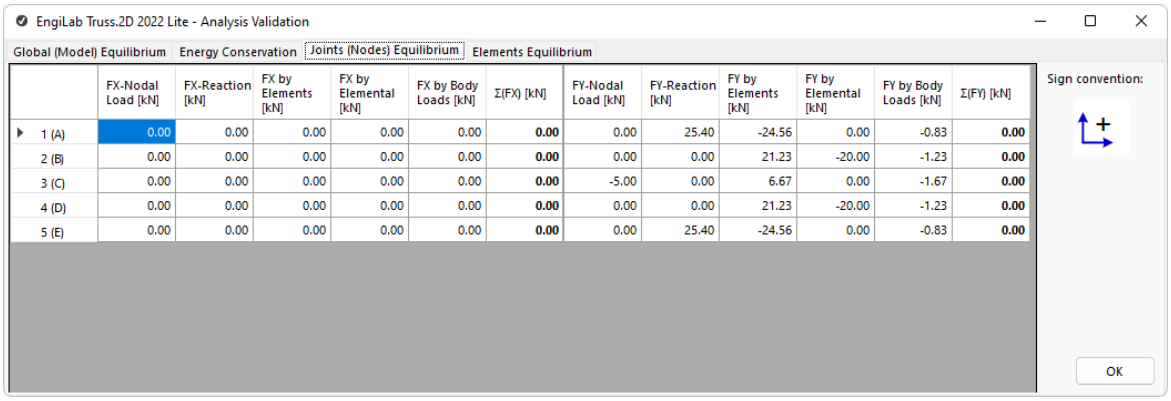
Work of Nodal Loads = 1/2*F*u,

where F is the Nodal Load and u is the displacement of the corresponding DOF from the Analysis. The reason why the factor 1/2 is used here is that the static force is applied slowly, increasing gradually from the value of zero to the F value (just like in a spring). So the work made by the external Force is not F*u, but rather 1/2*F*u.

(c) 2022 EngiLab PC

4.7.3 Joints (Nodes) Equilibrium

This tab shows the calculations related to the **Equilibrium of Joints (Nodes)**. After the Analysis, every joint of the 2D Truss needs to be in equilibrium state, i.e. the Sum of X-Forces and the Sum of Y-Forces need to be all zero for every joint of the Model. The program calculates all the Forces on each joint and also the Sums Σ(FX) and Σ(FY), as shown below.



	FX-Nodal Load [kN]	FX-Reaction [kN]	FX by Elements [kN]	FX by Elemental [kN]	FX by Body Loads [kN]	Σ(FX) [kN]	FY-Nodal Load [kN]	FY-Reaction [kN]	FY by Elements [kN]	FY by Elemental [kN]	FY by Body Loads [kN]	Σ(FY) [kN]
1 (A)	0.00	0.00	0.00	0.00	0.00	**0.00**	0.00	25.40	-24.56	0.00	-0.83	**0.00**
2 (B)	0.00	0.00	0.00	0.00	0.00	**0.00**	0.00	0.00	21.23	-20.00	-1.23	**0.00**
3 (C)	0.00	0.00	0.00	0.00	0.00	**0.00**	-5.00	0.00	6.67	0.00	-1.67	**0.00**
4 (D)	0.00	0.00	0.00	0.00	0.00	**0.00**	0.00	0.00	21.23	-20.00	-1.23	**0.00**
5 (E)	0.00	0.00	0.00	0.00	0.00	**0.00**	0.00	25.40	-24.56	0.00	-0.83	**0.00**

For this calculation, the following forces are taken into account for each direction, X and Y:

1. Any **Nodal Force** on the joint, i.e. the external Nodal Load. See the **1st column** for the X- and Y-directions in the figure above.
2. Any **Support Reaction Forces** from any constraints (supports) or springs on the Node. See the **2nd column** for the X- and Y-directions.
3. The **Forces contributed by Elements** that are "connected" to the Node. An Element with a tensile force F connected to a Node is "pulling" the Node with this Force along its direction. Similarly, an Element with a compressive force F connected to a Node is "pushing" the Node with this Force along its direction. The **3rd column** for the X- and Y-directions include all these contributions from any Elements connected to every Node.
4. The **Equivalent Nodal Forces due to Elemental Loads** on the Elements "connected" to the Node. An Element with a distributed load will contribute half of the load to its Start Node i and half of the load to its End Node j. See the **4th column** for the X- and Y-directions.
5. Similarly, the **Equivalent Nodal Forces due to Body Loads** on the Elements "connected" to the Node. An Element with a Body (Acceleration) load will contribute half of the load to its Start Node i and half of the load to its End Node j. See the **5th column** for the X- and Y-directions.

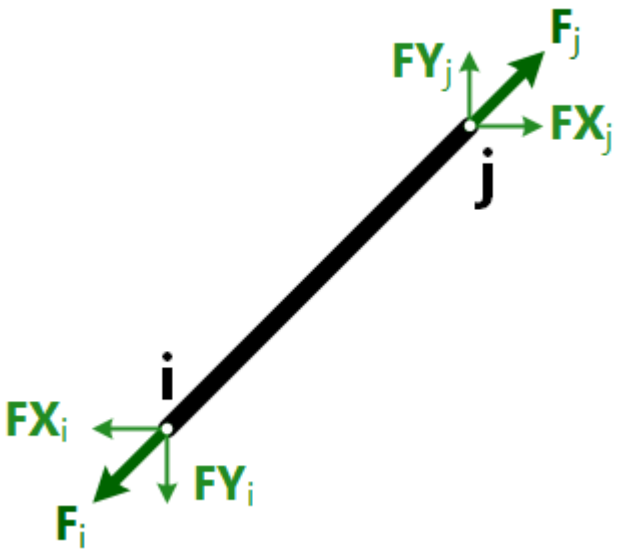
(c) 2022 EngiLab PC

The **6th column** for the X- and Y-directions in the figure above is the Sum of all the 5 forces on every Node. All the values on the 6th column for each direction have to be zero so that all joints are in equilibrium.

4.7.4 Elements Equilibrium

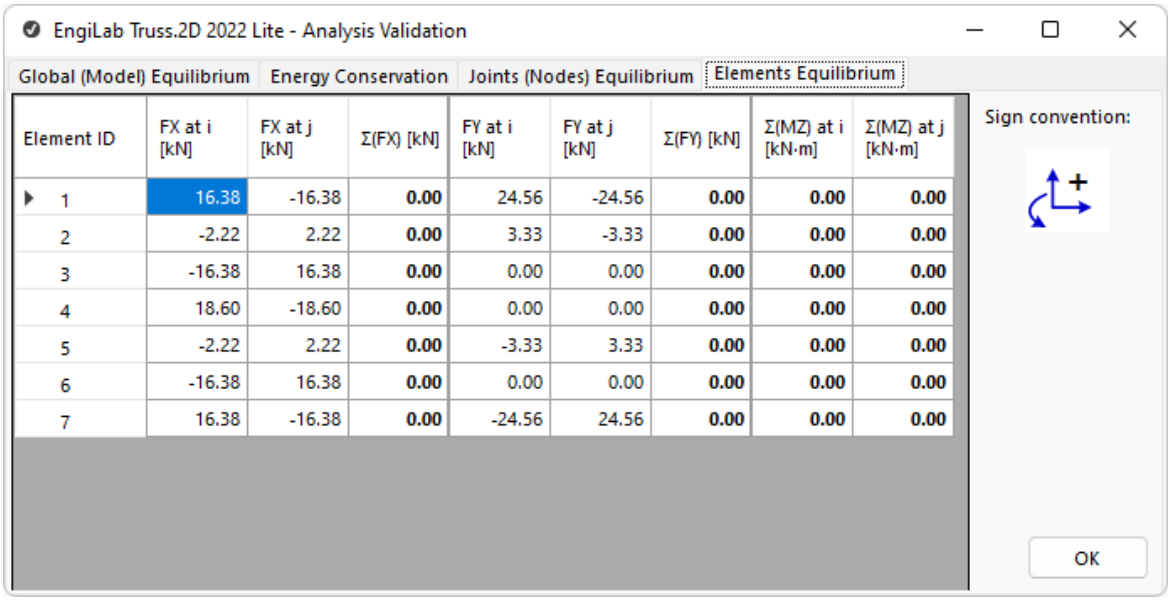
This tab shows the calculations related to the **Equilibrium of Elements**. After the Analysis, every Element of the 2D Truss needs to be in equilibrium state, i.e. the Sum of X-Forces and the Sum of Y-Forces need to be all zero for every Element. In addition, the Sum of moments of all Forces acting on the Element needs to be zero, at any point on the 2D plane.

Of course, any 2D Truss member which has only Axial Forces (no shear, no bending) and the Axial Force at Start i is the same as the Axial Force at End j (with opposite directions), satisfies these conditions, as a two-force body. See the illustration below where the Axial Forces F_i at i and F_j at j are decomposed into their two FX and FY components.



(c) 2022 EngiLab PC

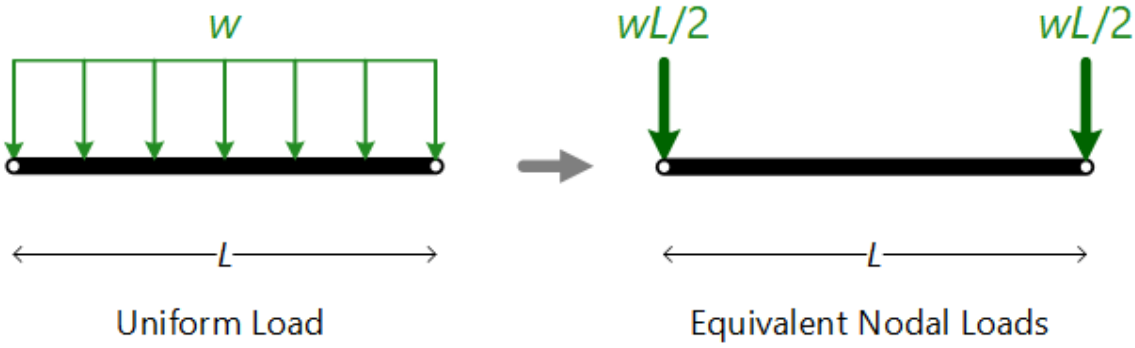
The program calculates these two components, the FX and FY Forces (in Global Axes) at both ends of each element, and the relevant Moments of these forces at i and j, as shown below.



Element ID	FX at i [kN]	FX at j [kN]	Σ(FX) [kN]	FY at i [kN]	FY at j [kN]	Σ(FY) [kN]	Σ(MZ) at i [kN·m]	Σ(MZ) at j [kN·m]
1	16.38	-16.38	0.00	24.56	-24.56	0.00	0.00	0.00
2	-2.22	2.22	0.00	3.33	-3.33	0.00	0.00	0.00
3	-16.38	16.38	0.00	0.00	0.00	0.00	0.00	0.00
4	18.60	-18.60	0.00	0.00	0.00	0.00	0.00	0.00
5	-2.22	2.22	0.00	-3.33	3.33	0.00	0.00	0.00
6	-16.38	16.38	0.00	0.00	0.00	0.00	0.00	0.00
7	16.38	-16.38	0.00	-24.56	24.56	0.00	0.00	0.00

In a 2D Truss, Elements do not have any loads (uniform or anything else) really applied on them. All loads are applied on the Nodes of the structure. In EngiLab Truss.2D, the user can apply self-weight of the members or other distributed loads (such as wind, snow and inertial forces), but **the program automatically converts these loads to equivalent Nodal Loads**. In the end, for the Analysis needs, there will be only Nodal Loads on the structure. Doing this, any effects of bending or shear on the Elements are neglected. The approach is explained below.

Assuming that a 2D Truss member carries a uniform load w (for example, in kN/m units), the total load is F=w*L (in kN), where L (in m, in this example) is the Length of the Element. For the Analysis, each Node (start and end Node) of this members takes a Force equal to F/2=w*L/2. This is shown schematically in the figure.



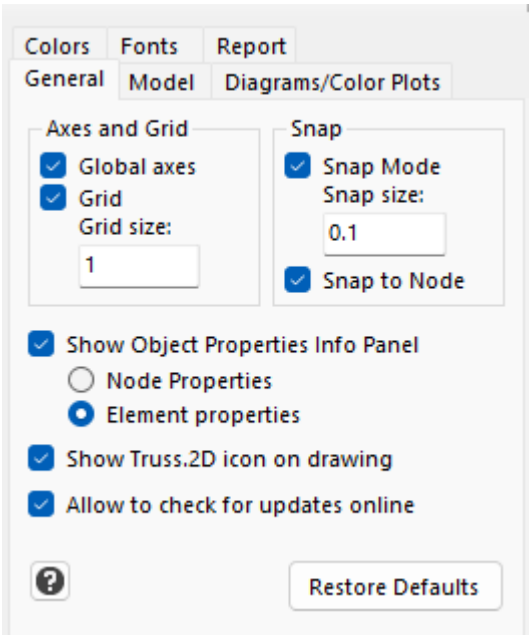
Chapter



Settings

5 Settings

5.1 General Settings



Available settings:

- **Global Axes**: Shows/Hides the Global Axes
- **Grid**: Shows/Hides the Grid
- **Grid size**: Sets the size of the Grid shown on screen

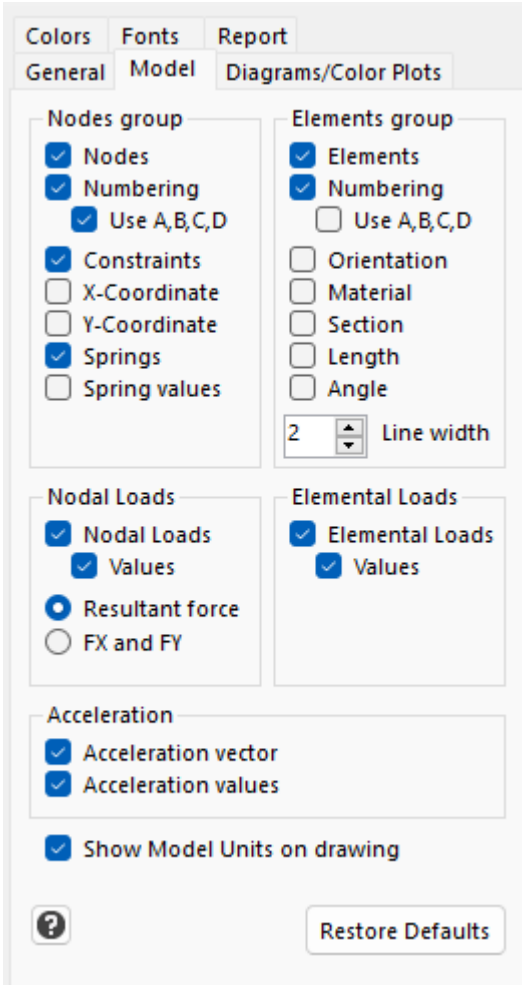
- **Snap Mode**: Enables/Disables the Snap Mode. If Snap Mode is enabled (Default=True), then you can only draw Nodes and Elements at increments defined by the Snap Size setting
- **Snap Size**: Sets the size of the Snap (Default=0.1)
- **Snap to Node**: Enables/Disables the Snap to Node. If Snap to Node is enabled (Default=True), then the user can "catch" Nodes on screen so that new elements can be connected to existing Nodes.

- **Show Object Properties Info Panel**: Shows/Hides the Object Properties Information Panel. It is used to show to the user the properties of Nodes and Elements highlighted on the screen with hovering over the Node or the Element.
- **Node properties** / **Element properties**: The user selects if Node or Element properties will be shown on the Information Panel.

(c) 2022 EngiLab PC

- **Show Truss.2D icon on drawing**: Shows/Hides the Truss.2D ($\mathbf{T_2}$) icon. When shown, it appears near the bottom right corner of the screen.
- **Allow to check for updates online**: Enables/Disables the automatic update feature. If disabled, the program will not check for updates online.

5.2 Model Settings



Available settings:

Nodes group

- **Nodes**: Shows/Hides the Nodes
- **Numbering**: Shows/Hides the Nodes' numbering
- **Use A,B,C,D**: If checked, the numbering of Nodes is not numerical but with the letters A, B, C, etc, like Excel columns numbering. This is useful to distinguish between the

(c) 2022 EngiLab PC

numbering of Nodes and Elements. By default, in EngiLab Truss.2D Nodes are numbered with letters A, B, C, and Elements are numbered with numbers, but the user can change these settings.

- **Constraints:** Shows/Hides the Constraints (Supports)
- **X-Coordinate:** Shows/Hides the X-Coordinate of the Node
- **Y-Coordinate:** Shows/Hides the Y-Coordinate of the Node
- **Springs:** Shows/Hides the Springs
- **Spring values:** Shows/Hides the Spring stiffness values

Elements group

- **Elements**: Shows/Hides the Elements (lines)
- **Numbering**: Shows/Hides the Elements' numbering
- **Use A,B,C,D**: If checked, the numbering of Elements is not numerical but with the letters A, B, C, etc, like Excel columns numbering.
- **Orientation:** Shows/Hides the Elements' orientation (arrow)
- **Material:** Shows/Hides the Elements' Material
- **Section:** Shows/Hides the Elements' Section
- **Length:** Shows/Hides the Elements' length
- **Angle:** Shows/Hides the Elements' angle
- **Line width:** The user sets the width of the Elements (lines) on screen

Nodal Loads

- **Nodal Loads**: Shows/Hides the Nodal Loads
- **Values**: Shows/Hides the values of the Nodal Loads
- **Resultant force** / **FX and FY**: In case of FX and FY Nodal force on a Node, the user selects if the Resultant force will be drawn (as a single vector) or of both vectors FX and FY will be drawn.

Elemental Loads

- **Elemental Loads**: Shows/Hides the Elemental Loads
- **Values**: Shows/Hides the values of the Elemental Loads

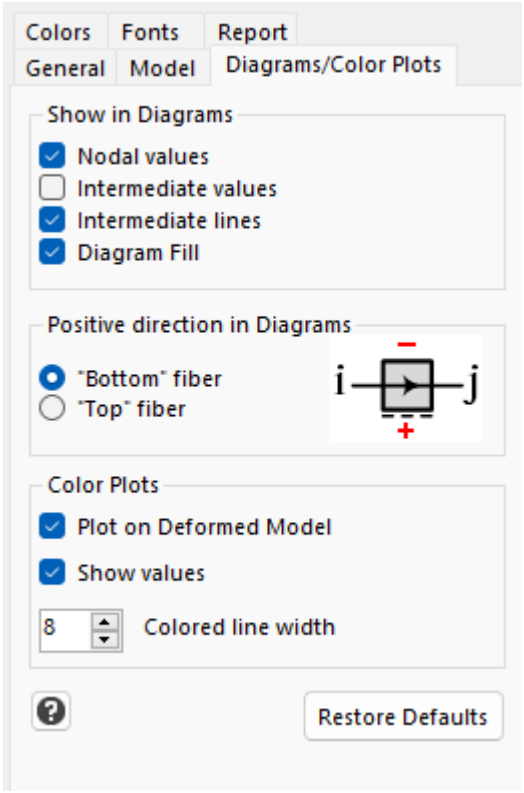
Acceleration

- **Acceleration vector**: Shows/Hides the Acceleration vector
- **Acceleration values**: Shows/Hides the Acceleration aX, aY values

(c) 2022 EngiLab PC

- **Show Model Units on drawing**: The user selects whether units will be shown on the drawing, for example if a force will appears as simply "10" or "10 kN/m" in case the load is 10 and the unit used is kN/m.

5.3 Axial Diagrams/Color Plots Settings



Available settings:

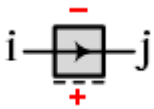
Show in Diagrams

- **Nodal values**: Shows/Hides the Nodal values of the Axial Diagrams (Force, Stress, Strain).
- **Intermediate values**: Shows/Hides the intermediate values of the Axial Diagrams.
- **Intermediate lines**: Shows/Hides the intermediate lines of the Axial Diagrams.
- **Diagram Fill**: Shows/Hides the Axial Diagrams fill.

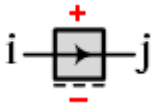
Positive direction in Diagrams (applies to all Axial Diagrams)

(c) 2022 EngiLab PC

Determines the positive direction of the diagrams. Applies to all three diagrams (Force, Stress, Strain). There are two options:



Positive towards the "Bottom" fiber (**Default**). Convention usually used in Europe.



Positive towards the "Top" fiber. Convention usually used in the US.

- **Nodal values**: Shows/Hides the Nodal values of the Axial Diagrams (Force, Stress, Strain).
- **Intermediate values**: Shows/Hides the intermediate values of the Axial Diagrams.

Color Plots

- **Plot on Deformed Model**: The user selects if the Color Plot will be drawn on the Deformed state of the Model or on the original Model.
- **Show values**: Shows/Hides the Axial Force/Stress/Strain values for every Element in the Color Plot.
- **Colored line width**: The width (thickness) of the colored line for every Element.

Note:

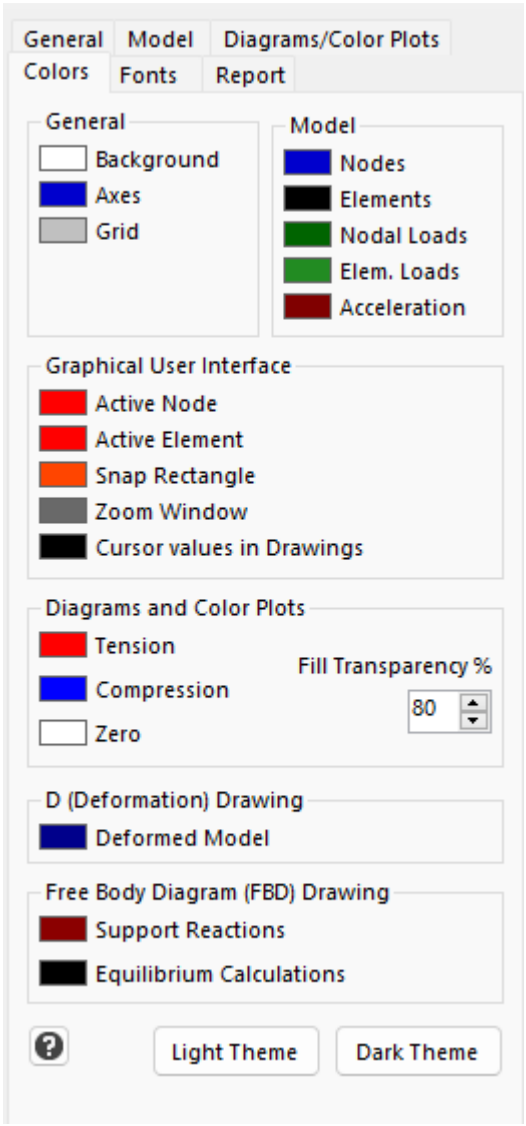
In EngiLab Truss.2D, the sign convention for the **Axial Force** (obviously also for Axial Stress and Axial Strain) is the following:



Axial Force is **Positive** when member is in **Tension**

(c) 2022 EngiLab PC

5.4 Colors Settings



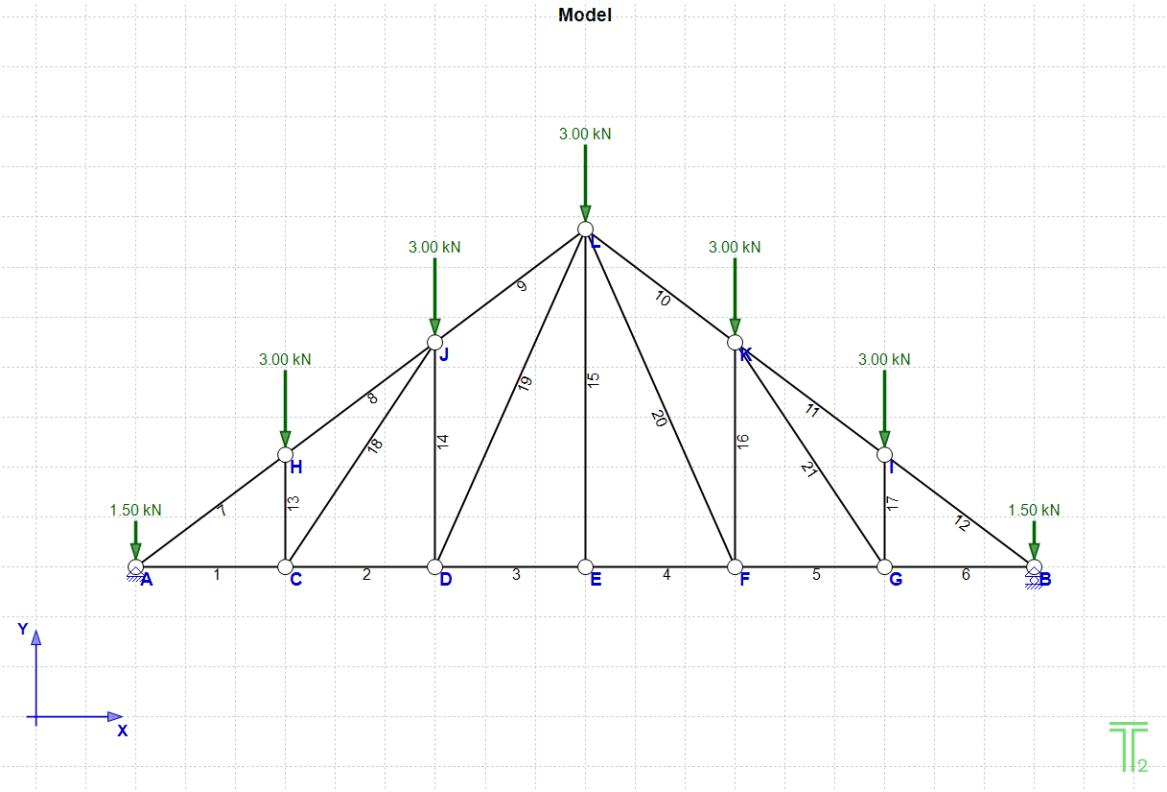
Each setting controls the color of the corresponding object on screen.

You can also set the **transparency** of the filled objects (for the Axial Force, Axial Stress and Axial Strain diagrams).

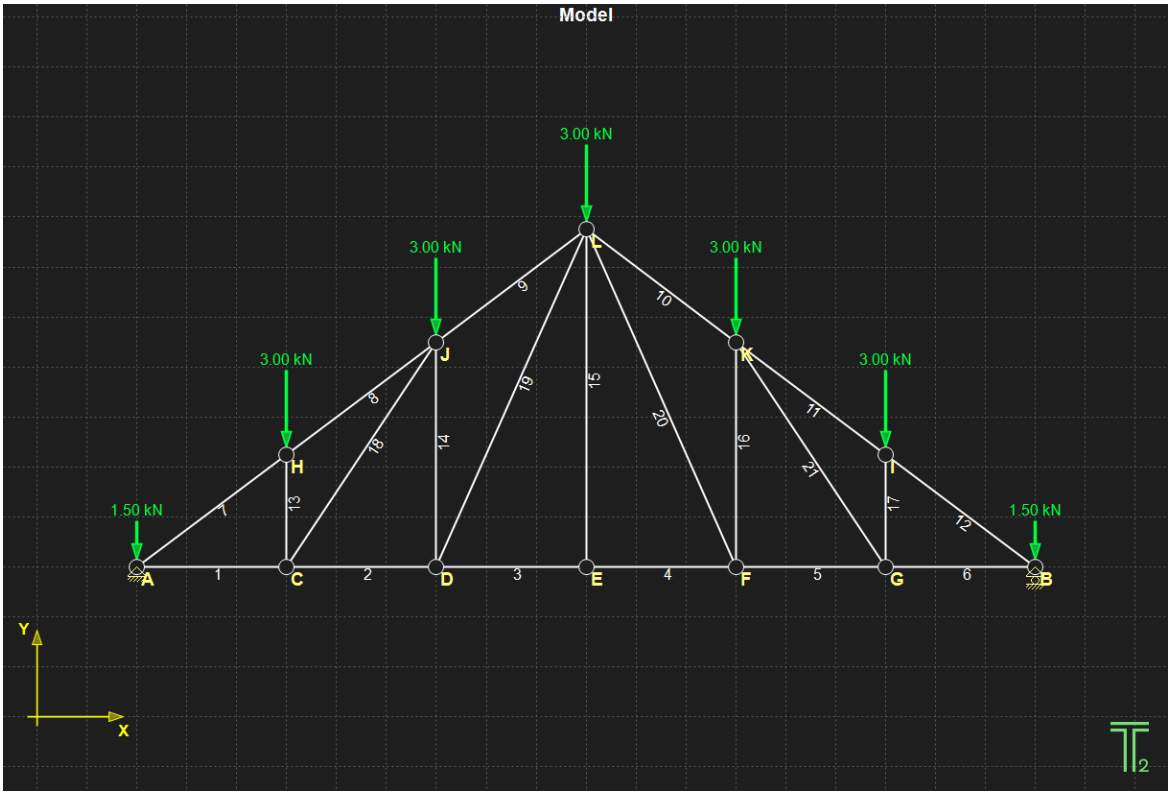
There are two predefined color themes available, the **Light Theme (Default)** and the **Dark Theme**. The two themes are shown below.

(c) 2022 EngiLab PC

Light Theme (Default)

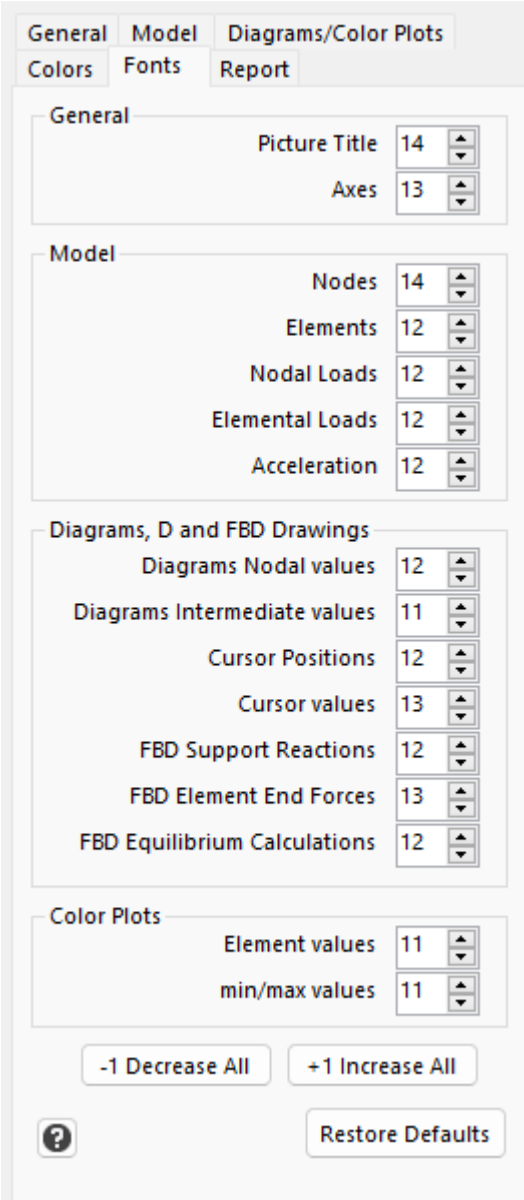


Dark Theme



(c) 2022 EngiLab PC

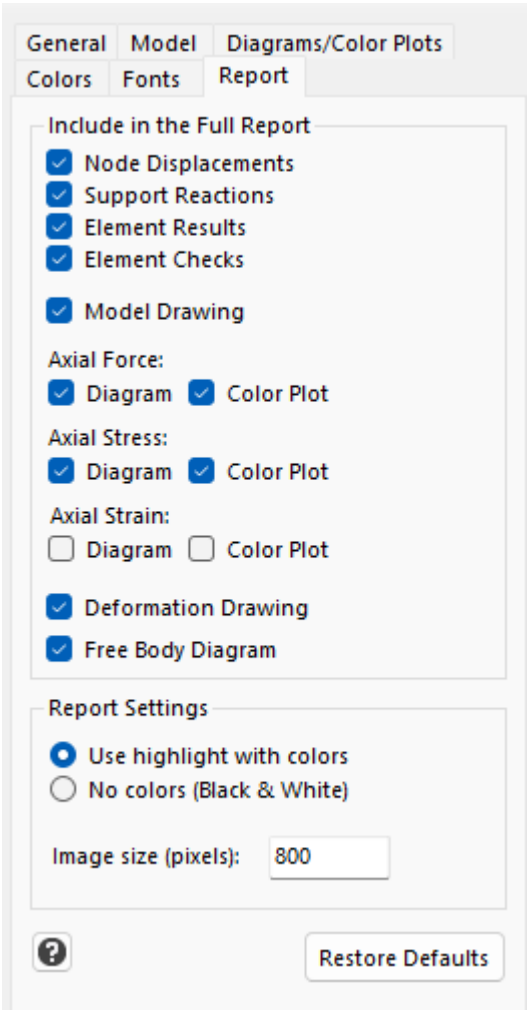
5.5 Fonts Settings



Each setting sets the Font size for the corresponding object on the screen.

(c) 2022 EngiLab PC

5.6 Report Settings



Available settings:

Include in the Full Report

The user selects what to include in the Full Report. By default the Axial Strain Diagram (and the Color Plot) are not included, while the Axial Force and Axial Stress Diagrams (and the corresponding Color Plots) are included.

Report Settings

- **Use highlight with colors / No colors (Black & White).** The user can choose to use colors to highlight special items (Constraints, Springs, etc) or only Black and White without highlighting special items.
- **Image size (pixels)**: The user can select the (max) size of the images, in pixels.

(c) 2022 EngiLab PC

This page intentionally left blank.

Chapter



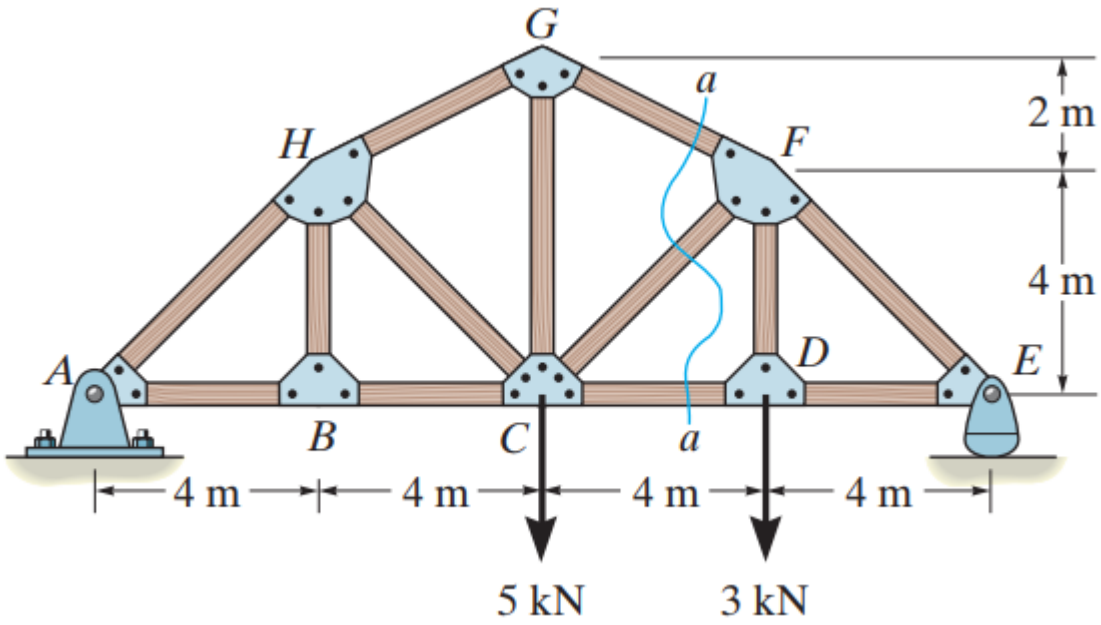
Example Problems

6 Example Problems

6.1 Example Problem 1

The first example is the 2D Truss shown in the figure. The figure and the problem are taken from the book "**Engineering Mechanics: Statics**" from **Russell C. Hibbeler** [1]. The problem, as described in [1] is stated as:

"Determine the **force in member CF** of the truss. Indicate whether the member is in tension or compression. Assume each member is pin connected."



Note that this is a **statically determinate 2D Truss**. In statically determinate trusses, all support reactions and member forces can be determined only by taking into account the equilibrium equations of the Model. This means that the Material Elastic Modulus (E) and the Section Area (A) properties of each member are not important in determining these quantities. Any E and A, for any member, will lead to the same results, when forces and support reactions are concerned. The same is not true for displacements. Displacements do depend on the Elastic Modulus and the Area of each member, but they are not asked for in this problem, at least in the book.

For our calculations in %PROGRAM NAME%>, we need to assign a Material and a Section to each Element, as the program is general and can analyze any 2D Truss, statically determinate or statically indeterminate, while it also calculates the displacements of all Nodes, among many others.

To draw a structure quickly without the need to set the Material and Section properties manually, EngiLab Truss.2D offers the "**QuickStart**" option, where the program asks the user if he/she agrees to assign a predefined Material and Section to each Element drawn on screen. The predefined Material and Section properties that EngiLab Truss.2D assigns to each member when drawn on screen are the following:

(c) 2022 EngiLab PC

Material

The predefined Material is "**Steel S 235**" with the following properties:

- Elastic Modulus **E = 210 GPa**
- Density **d = 7850 kg/m³**
- Yield Stress **fy = 235 MPa**

Section

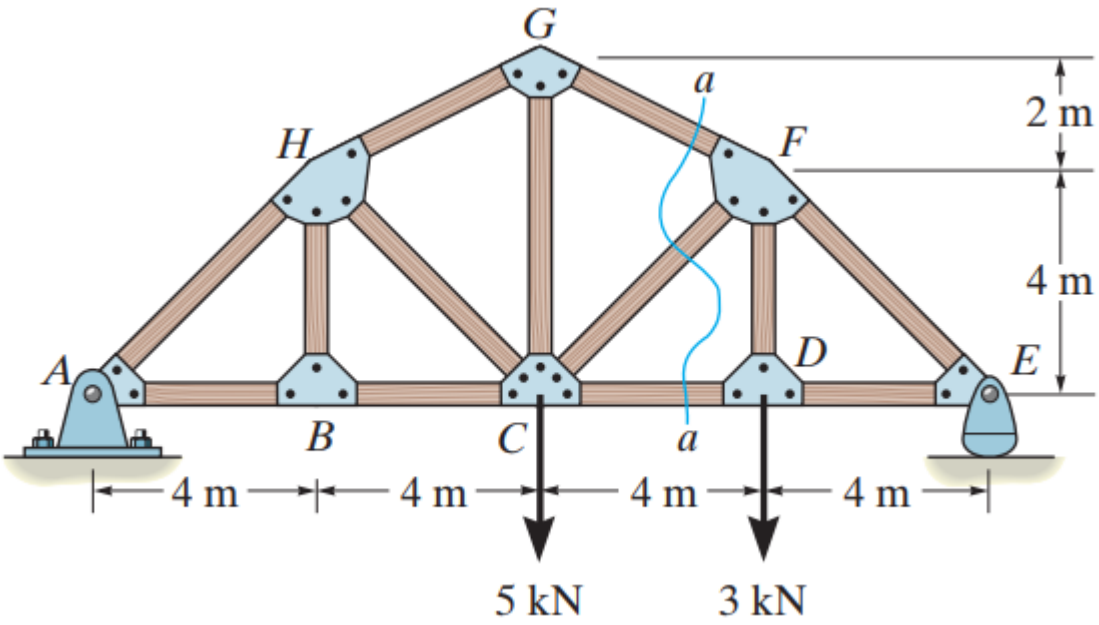
The predefined Section is "**IPE 200**" with the following properties:

- Area **A = 28.5 cm²**
- Moment of Inertia **I = 1943 cm⁴**

References

[1] Engineering Mechanics: Statics, R.C. Hibbeler, 14th Edition, Pearson Prentice Hall, ISBN-13: 978-0133918922, 2016.

6.1.1 Example 1 - Hand Calculations



The solution, from the book [1], is the following, using hand calculations:

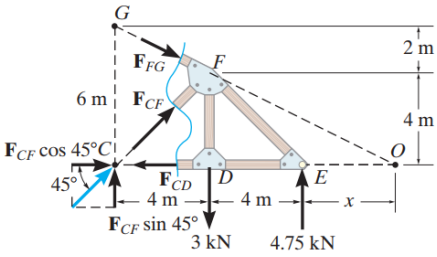
(c) 2022 EngiLab PC

SOLUTION

Free-Body Diagram. Section *aa* in Fig. 6–17*a* will be used since this section will "expose" the internal force in member *CF* as "external" on the free-body diagram of either the right or left portion of the truss. It is first necessary, however, to determine the support reactions on either the left or right side. Verify the results shown on the free-body diagram in Fig. 6–17*b*.

The free-body diagram of the right portion of the truss, which is the easiest to analyze, is shown in Fig. 6–17*c*. There are three unknowns, F_{FG}, F_{CF}, and F_{CD}.

Equations of Equilibrium. We will apply the moment equation about point *O* in order to eliminate the two unknowns F_{FG} and F_{CD}. The location of point *O* measured from *E* can be determined from proportional triangles, i.e., $4/(4 + x) = 6/(8 + x)$, $x = 4$ m. Or, stated in another manner, the slope of member *GF* has a drop of 2 m to a horizontal distance of 4 m. Since *FD* is 4 m, Fig. 6–17*c*, then from *D* to *O* the distance must be 8 m.



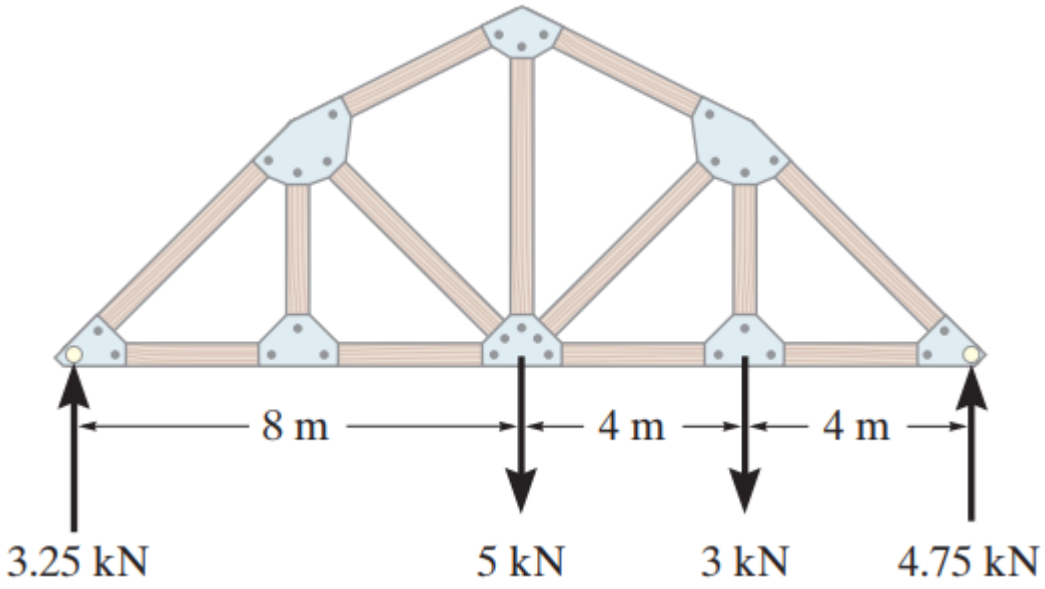
(c)

An easy way to determine the moment of $\mathbf{F}_{CF}$ about point *O* is to use the principle of transmissibility and slide $\mathbf{F}_{CF}$ to point *C*, and then resolve $\mathbf{F}_{CF}$ into its two rectangular components. We have

$$\zeta + \Sigma M_O = 0;$$

$$-F_{CF}\sin 45°(12\text{ m}) + (3\text{ kN})(8\text{ m}) - (4.75\text{ kN})(4\text{ m}) = 0$$

$$F_{CF} = 0.589\text{ kN} \quad (C) \qquad \textit{Ans.}$$

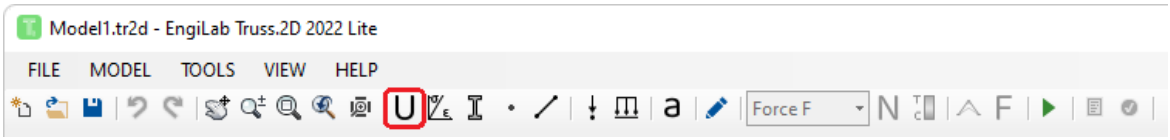


References

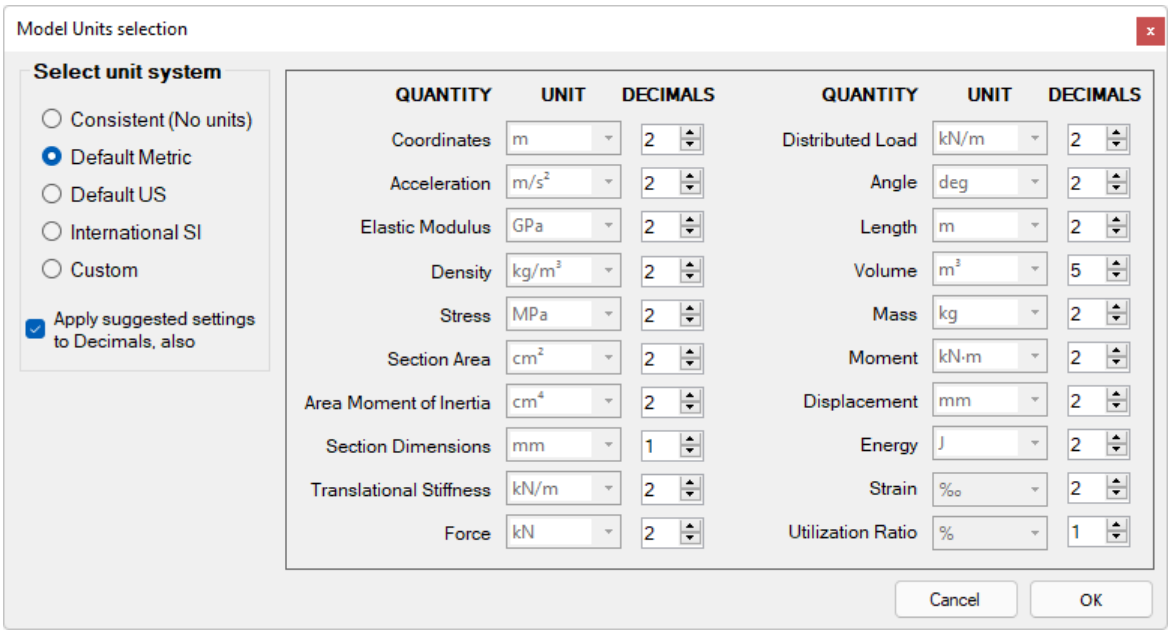
[1] Engineering Mechanics: Statics, R.C. Hibbeler, 14th Edition, Pearson Prentice Hall, ISBN-13: 978-0133918922, 2016.

(c) 2022 EngiLab PC

6.1.2 Step 1. Define the System of Units

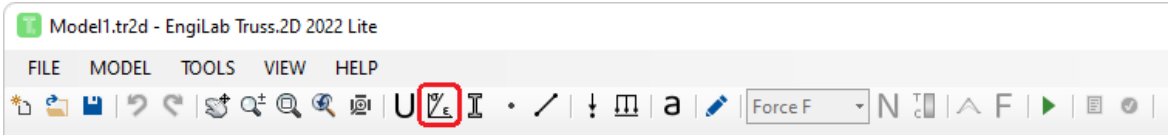


Click U to define the **System of Units**. This is a very important step for the Model. In this example, we will use the **Default Metric** System of units. For details, see Units



Select the **Default Metric** System of Units. Click OK.

6.1.3 Step 2. Define Materials



In our example, we will use the **QuickStart** feature of the program and there is no need to define the Material manually. **You can go directly to Step 4. Draw the Model on screen**.

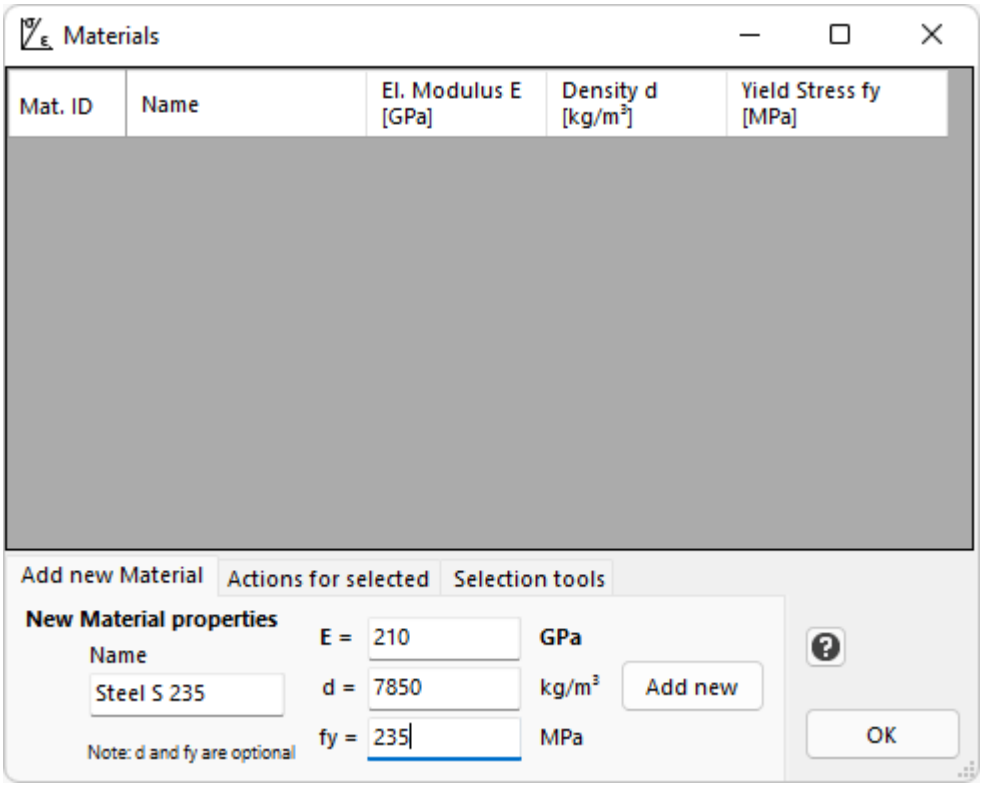
If on the other hand, you prefer to define the Material manually, then you can follow the instructions on this page.

(c) 2022 EngiLab PC

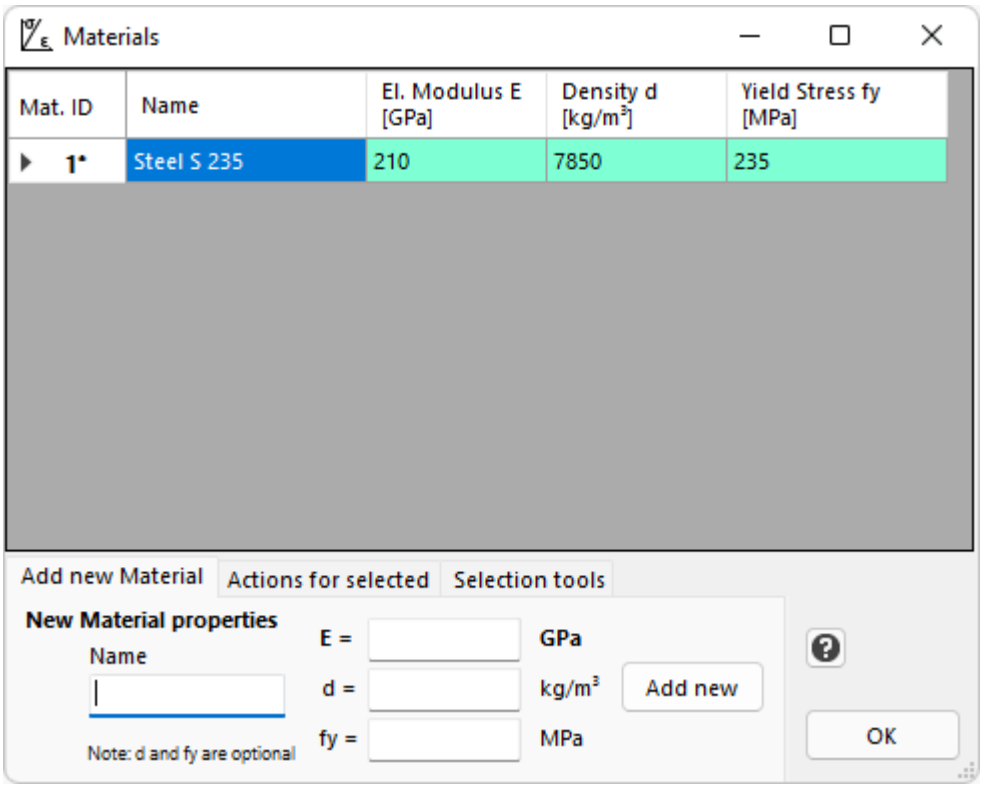
Click to define **Material**(s). For details, see Materials.

You need to define the Material **Name** (optional), **Material Density d** (optional in general, only needed if you are to define self-weight loads, which is not the case here) and Material **Elastic Modulus E** for each Material. Use the following properties:

- Name: **Steel S 235**
- Elastic Modulus **E = 210 GPa**
- Density **d = 7850 kg/m³**
- Yield Stress **fy = 235 MPa**

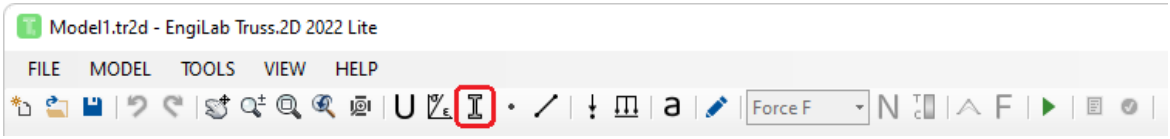


Then **Click the "Add new" button**. The new Material will be added to the table that shows the Model Materials.



Then **Click OK to exit the Materials form**.

6.1.4 Step 3. Define Sections



In our example, we will use the **QuickStart** feature of the program and there is no need to define the Section manually. **You can go directly to Step 4. Draw the Model on screen**.

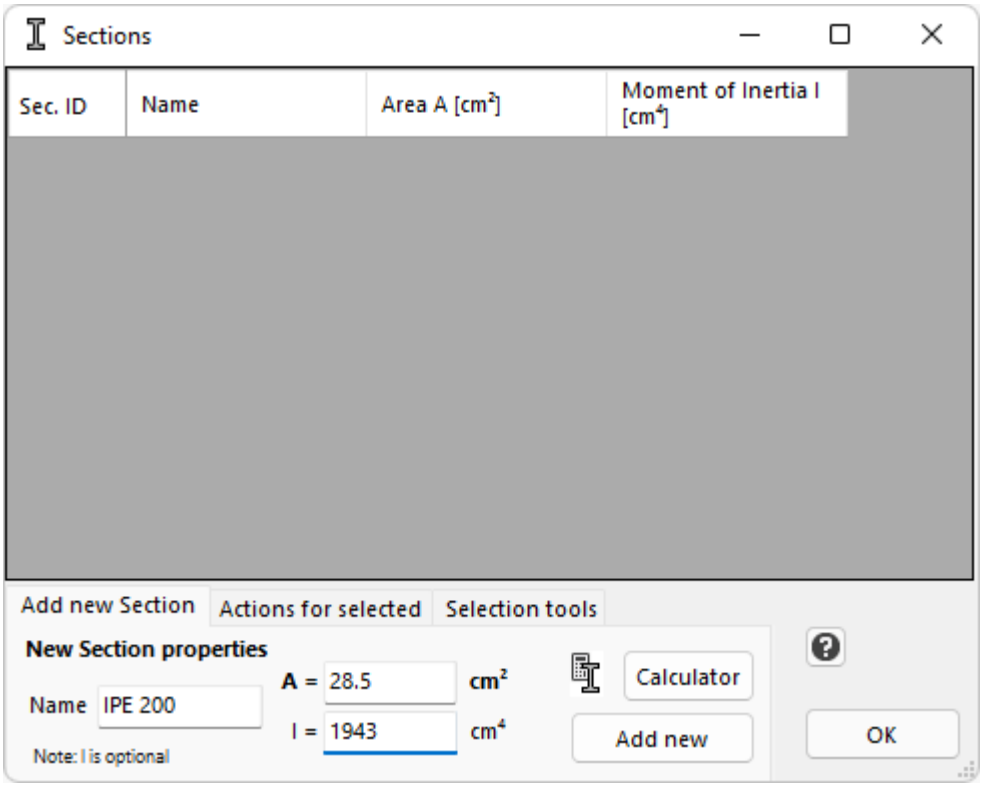
If on the other hand, you prefer to define the Section manually, then you can follow the instructions on this page.

Click I to define **Section**(s). For details, see Sections.

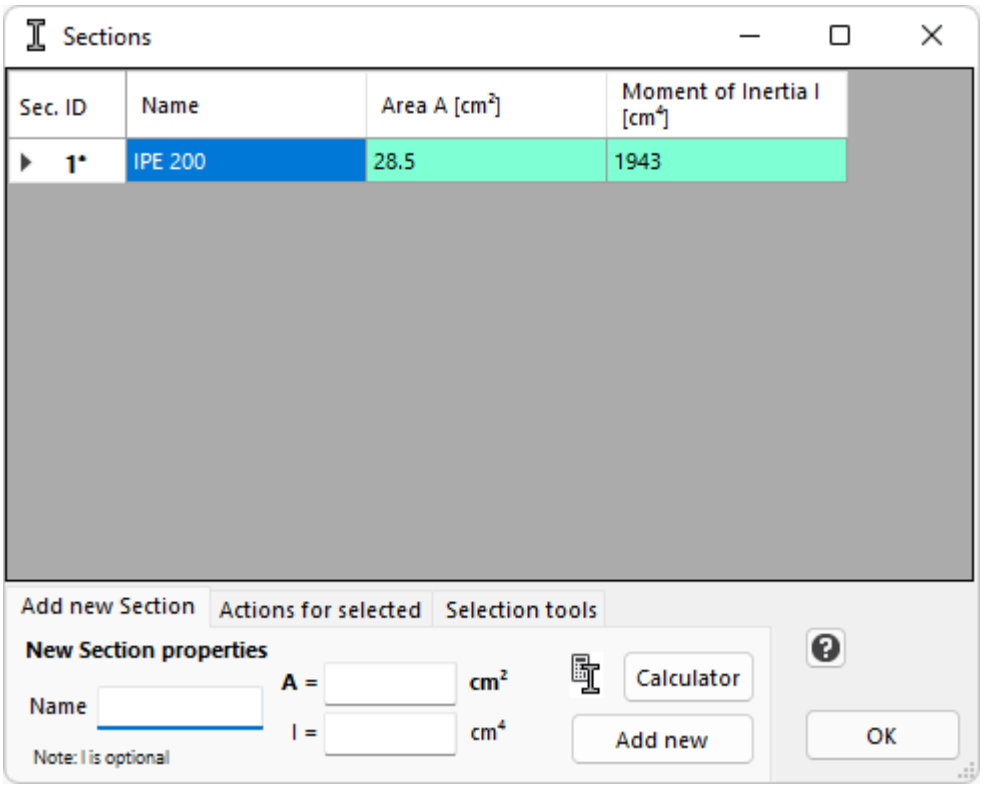
(c) 2022 EngiLab PC

You need to define the Section **Name** (optional), **Section Area A** and **Section Moment of Inertia I** (optional in general, only needed if you want to check the Elements for buckling, which is not the case here). Use the following properties:

- Name: **IPE 200**
- Area **A = 28.5 cm²**
- Moment of Inertia **I = 1943 cm⁴**



Then **Click the "Add new" button**. The new Section will be added to the table that shows the Model Sections.



Then **Click OK to exit the Sections form**.

6.1.5 Step 4. Draw the Model on screen

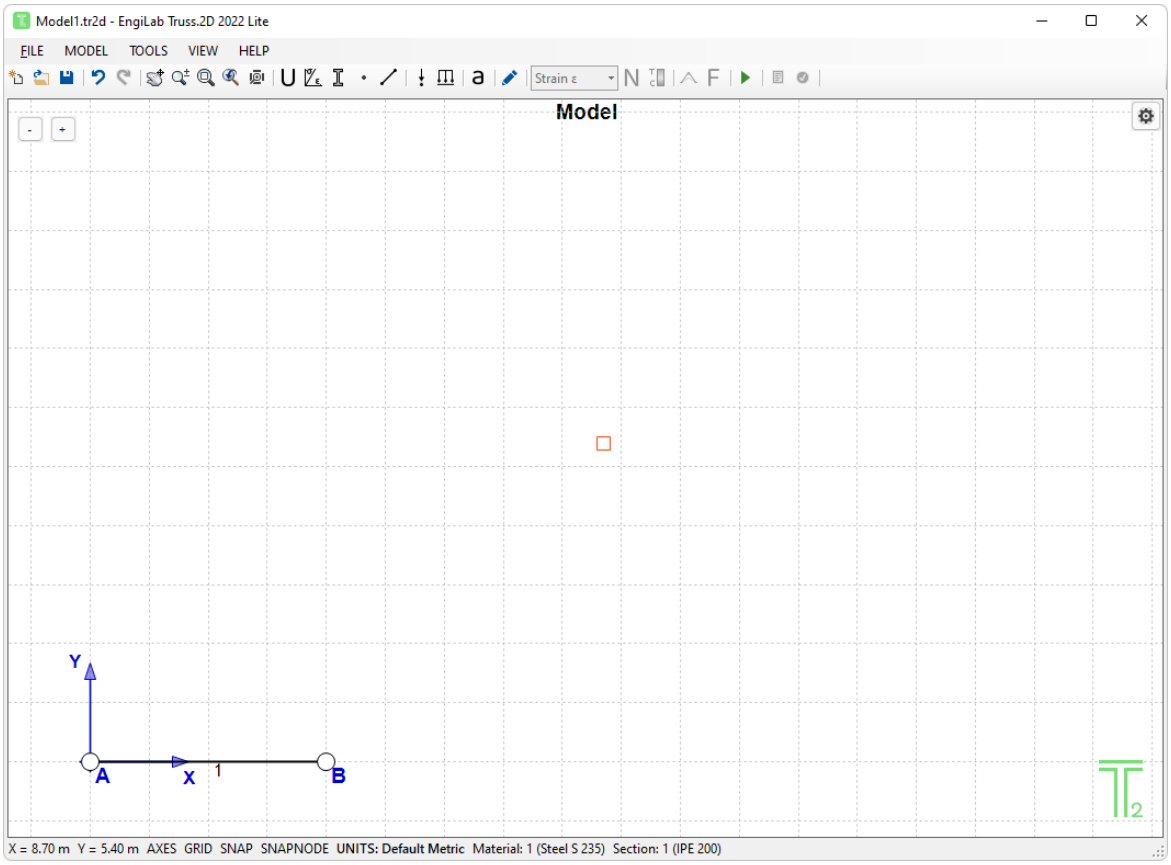
After you have defined the Material and the Section, you can **start drawing your Model on the screen**.

- First make sure that **SNAP** and **SNAPNODE** are both activated (see bottom left of Picture below).
- If **SNAP** is activated (Default=True), then you can only draw Nodes and Elements at increments defined by the Snap Size setting (Default=0.1). This is fine for our example.
- If **SNAPNODE** is activated (Default=True), then you can "catch" Nodes so that new elements can be connected to existing Nodes. This is essential for building our Model.
- Left-click on screen, hold down the left button and then release it at another location to define a new Element and two Nodes at ends i, j.
- **Note**: **All Elements that are defined on screen are assigned the Default Material and the Default Section 1**. If there is only one Material or Section, then this is the Default Material or Section, otherwise the user can select among the different Materials or Sections.

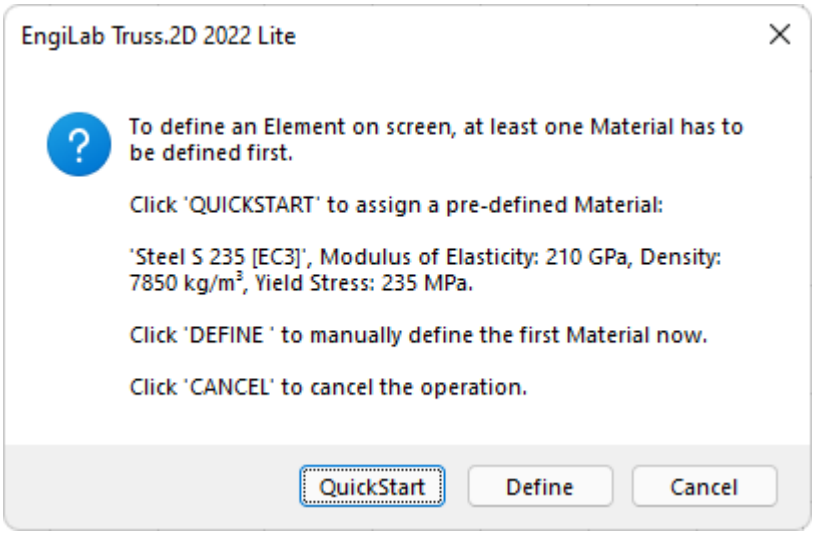
(c) 2022 EngiLab PC

You can start at any point. Let's start from point 0,0 (Origin of axes). Draw the first member as shown below. Starting from point **(0, 0)**, release the left button after you have reached the **(4, 0)** point on screen. Coordinates are shown on the status bar at the bottom left corner of the window.

A. In case you have defined the Material and the Section manually (you have followed Steps 3 and 4), you will get the following screen:

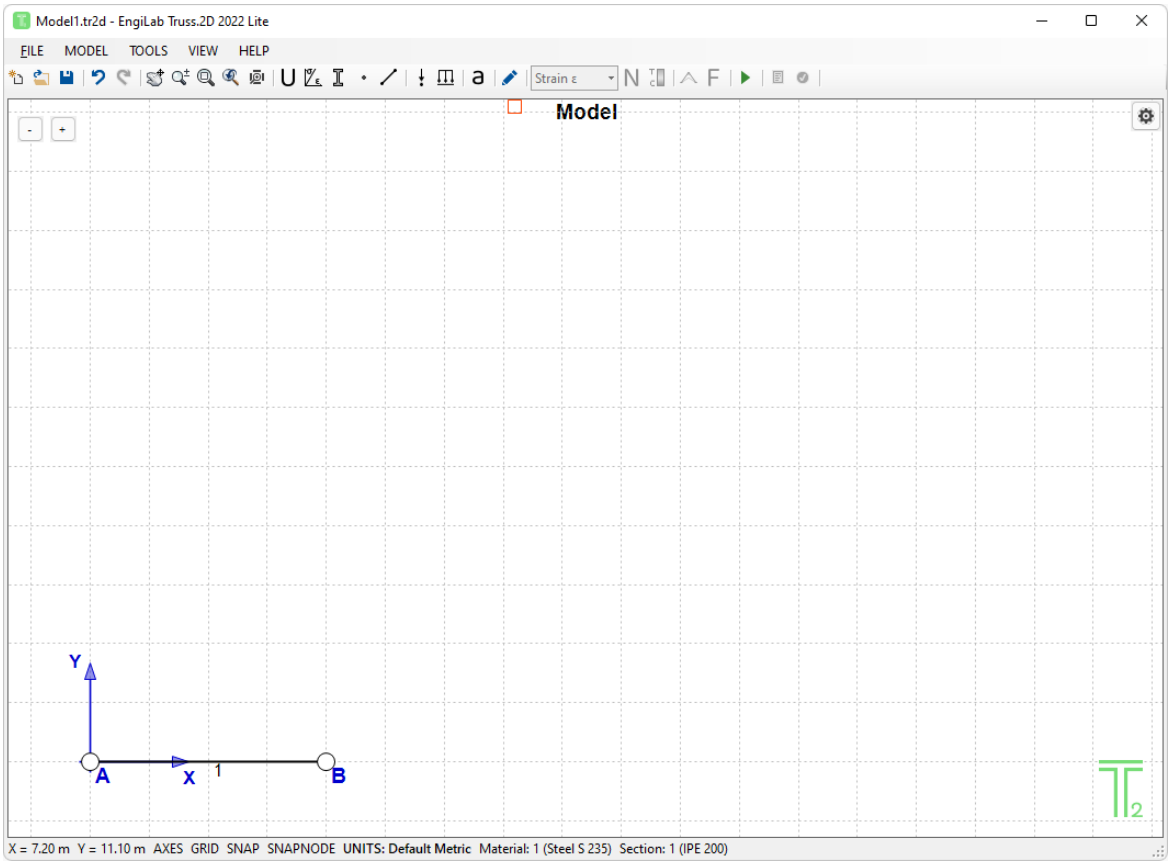


B. In case you have NOT defined the Material and the Section manually (you have skipped Steps 3 and 4), you will get the following prompt:



(c) 2022 EngiLab PC

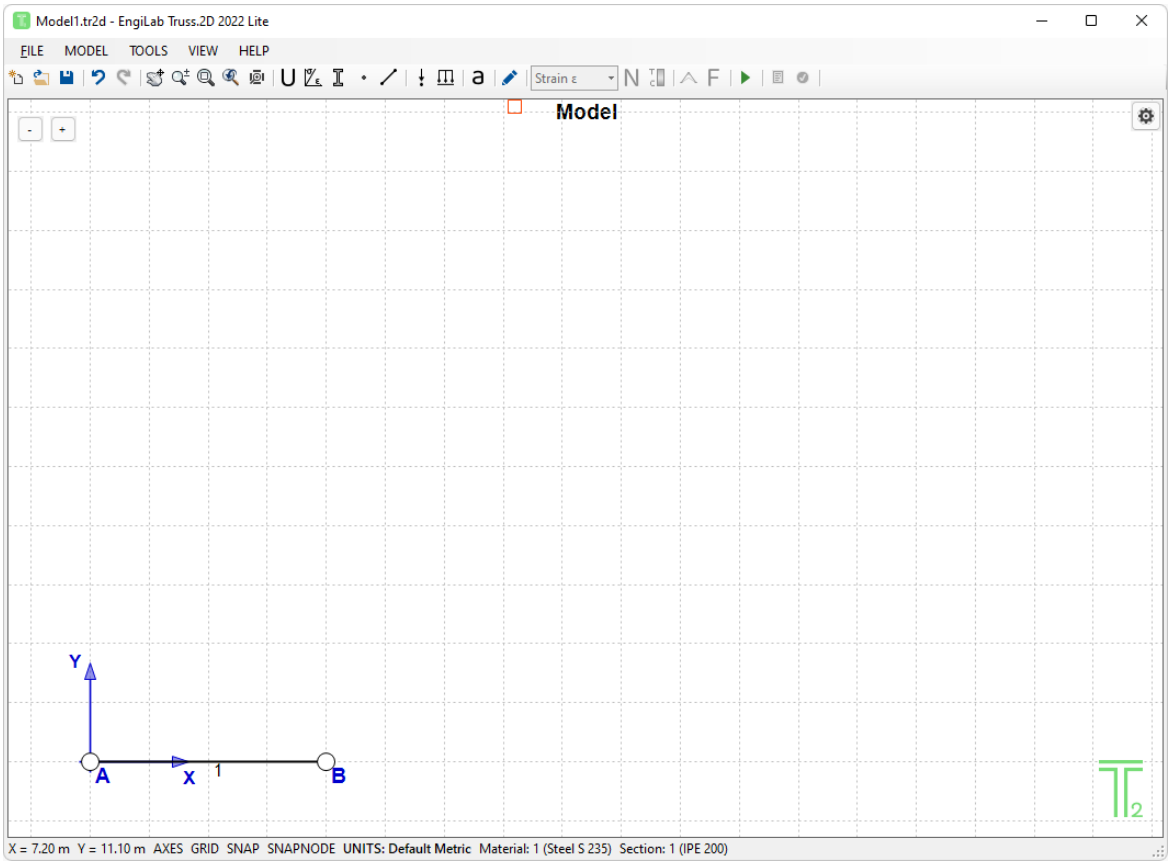
Click QuickStart to quickly assign the predefined **Material**. You get another prompt:



Click QuickStart again to quickly assign the predefined **Section** and start drawing.

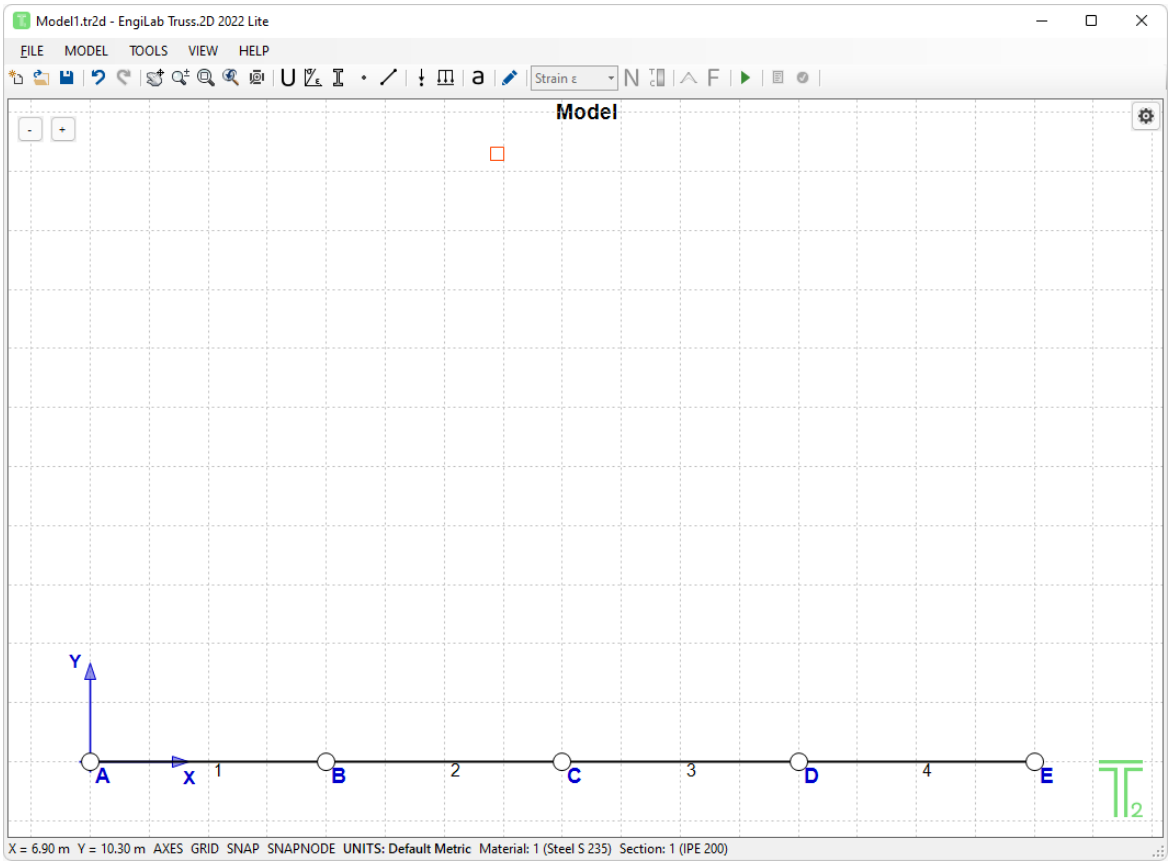
In both cases, you will finally get the first Element and the first two Nodes as follows:

(c) 2022 EngiLab PC

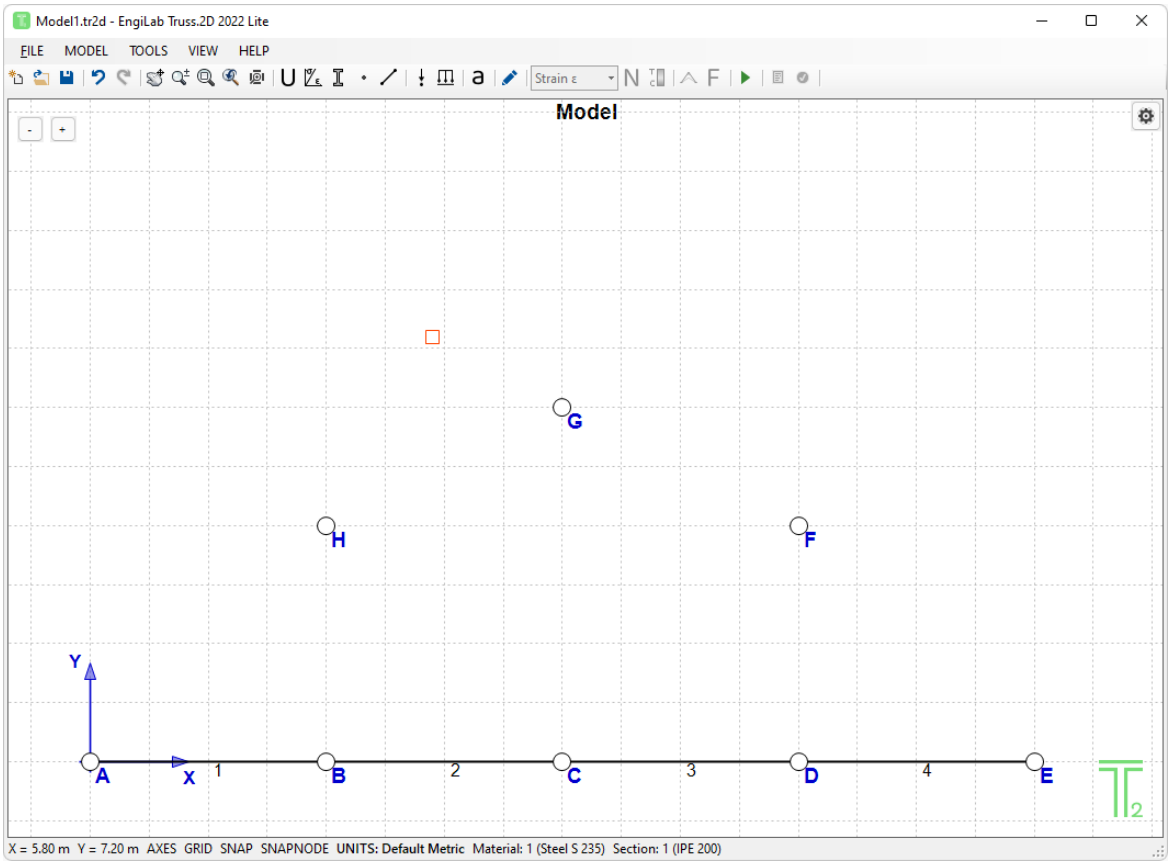


Using the same technique, continue with the other horizontal Elements, as shown in the figure below.

(c) 2022 EngiLab PC

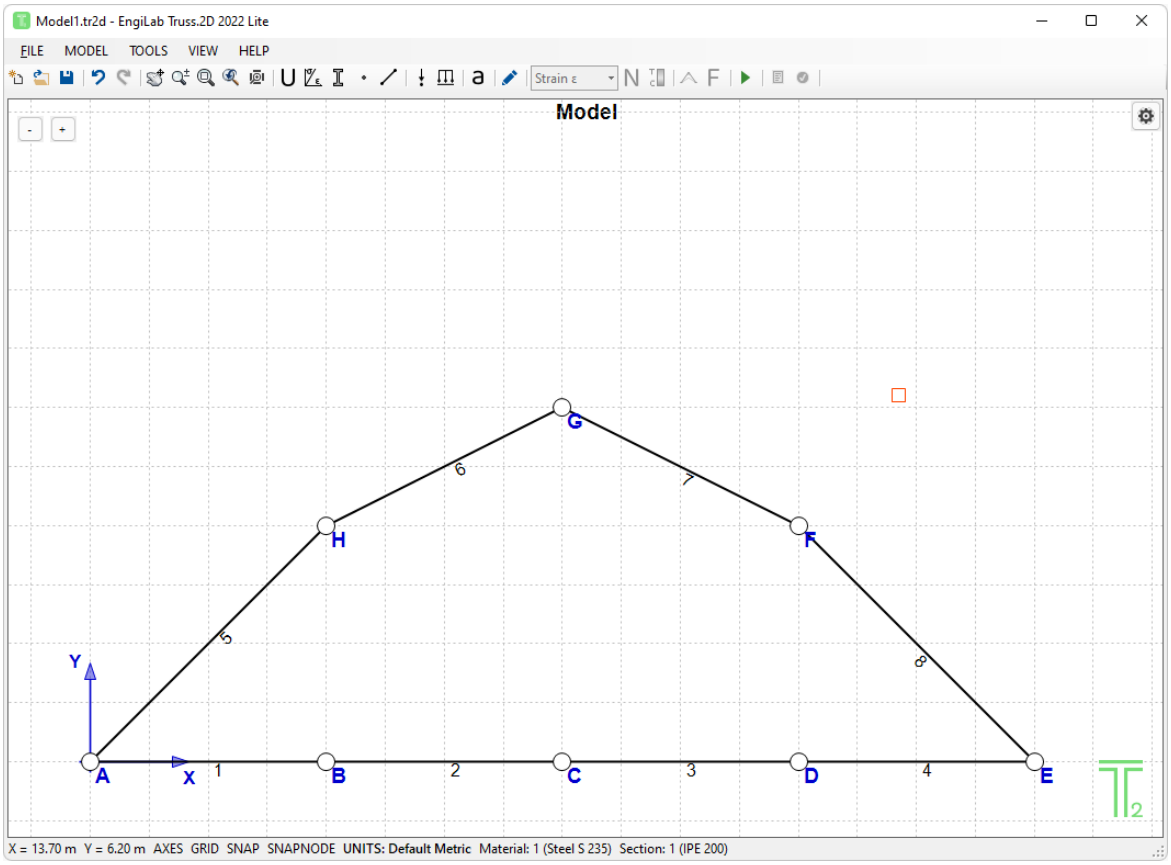

Model1.tr2d - EngiLab Truss.2D 2022 Lite
FILE
MODEL
TOOLS
VIEW
HELP
Strain ε
Model
Y
X
A
1
B
2
C
3
D
4
E
X = 6.90 m Y = 10.30 m AXES GRID SNAP SNAPNODE UNITS: Default Metric Material: 1 (Steel S 235) Section: 1 (IPE 200)

Now draw the three other Nodes, as shown in the figure below.

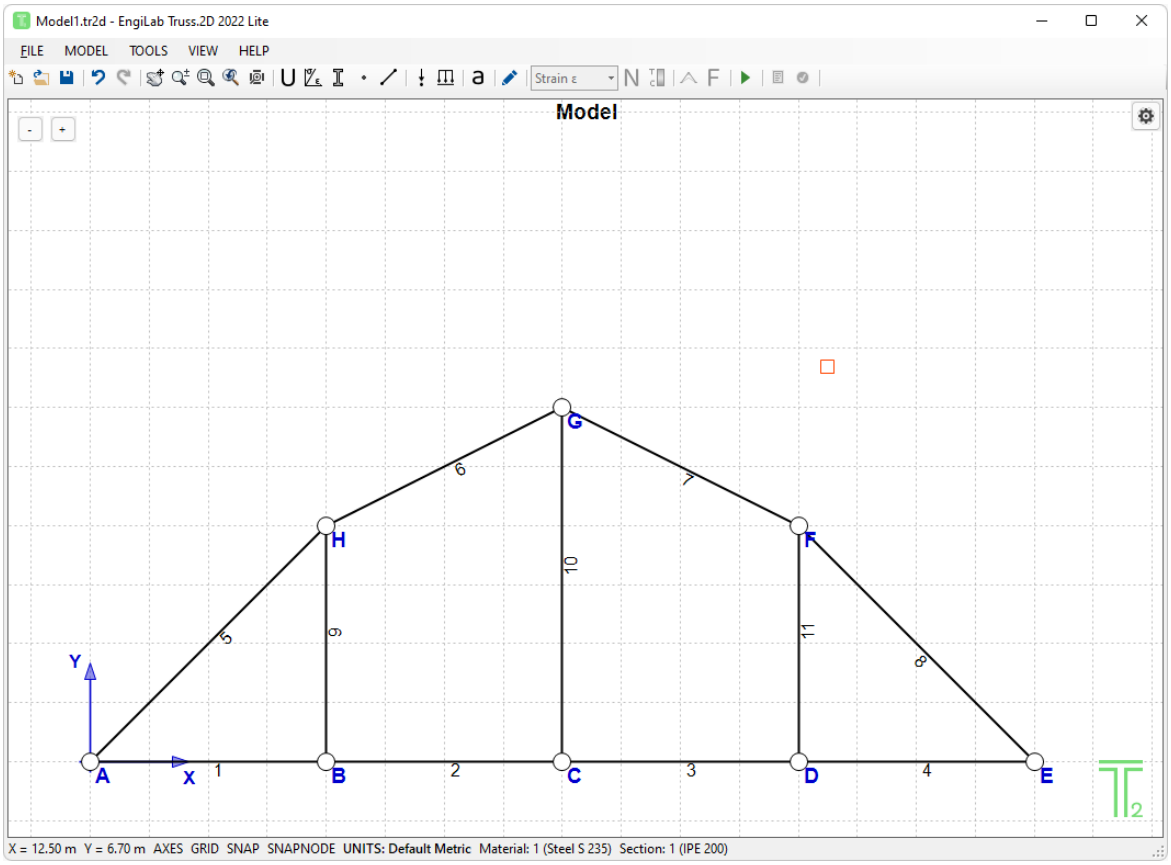


(c) 2022 EngiLab PC

And connect them with Elements, as shown below.

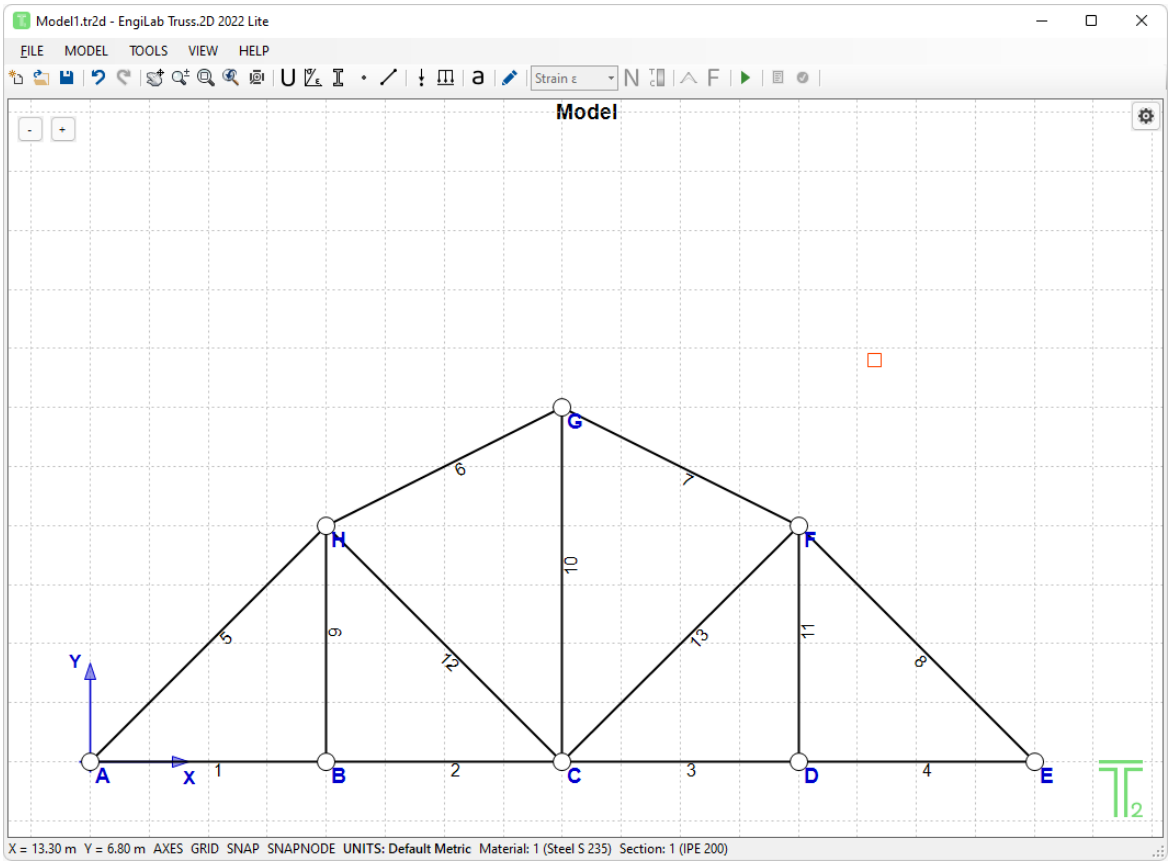


Continue with the vertical Elements.



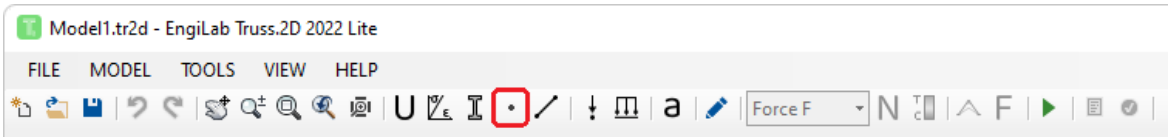
Finally, draw the 2 diagonal Elements.

(c) 2022 EngiLab PC



That's it! Now you have built the basic Model. You still need to apply the Supports and the external loading.

6.1.6 Step 5. Edit Nodes

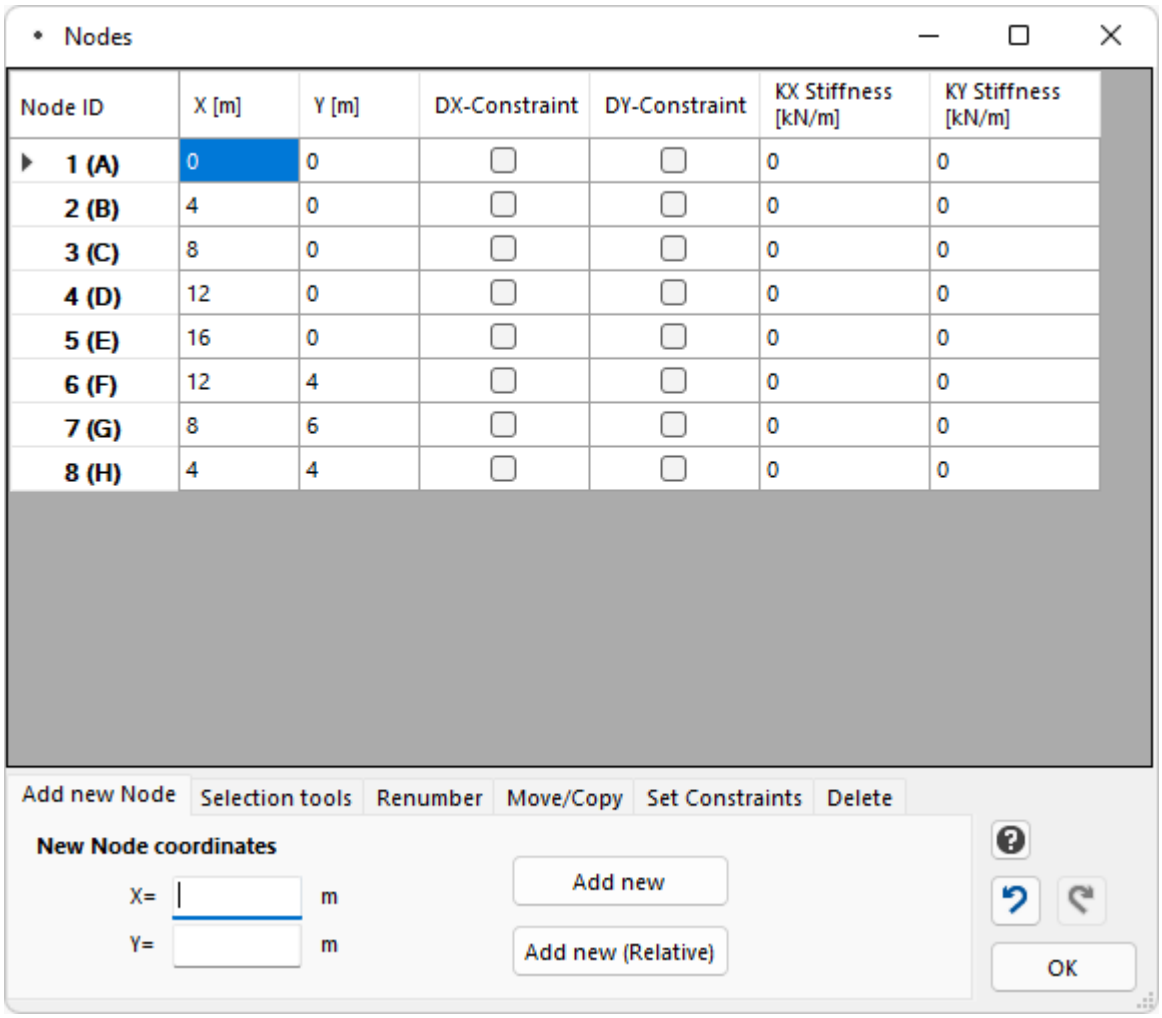


After the preparation of the Model on screen, you need to do some corrections on the Nodes that have been generated.

For example you may want to:

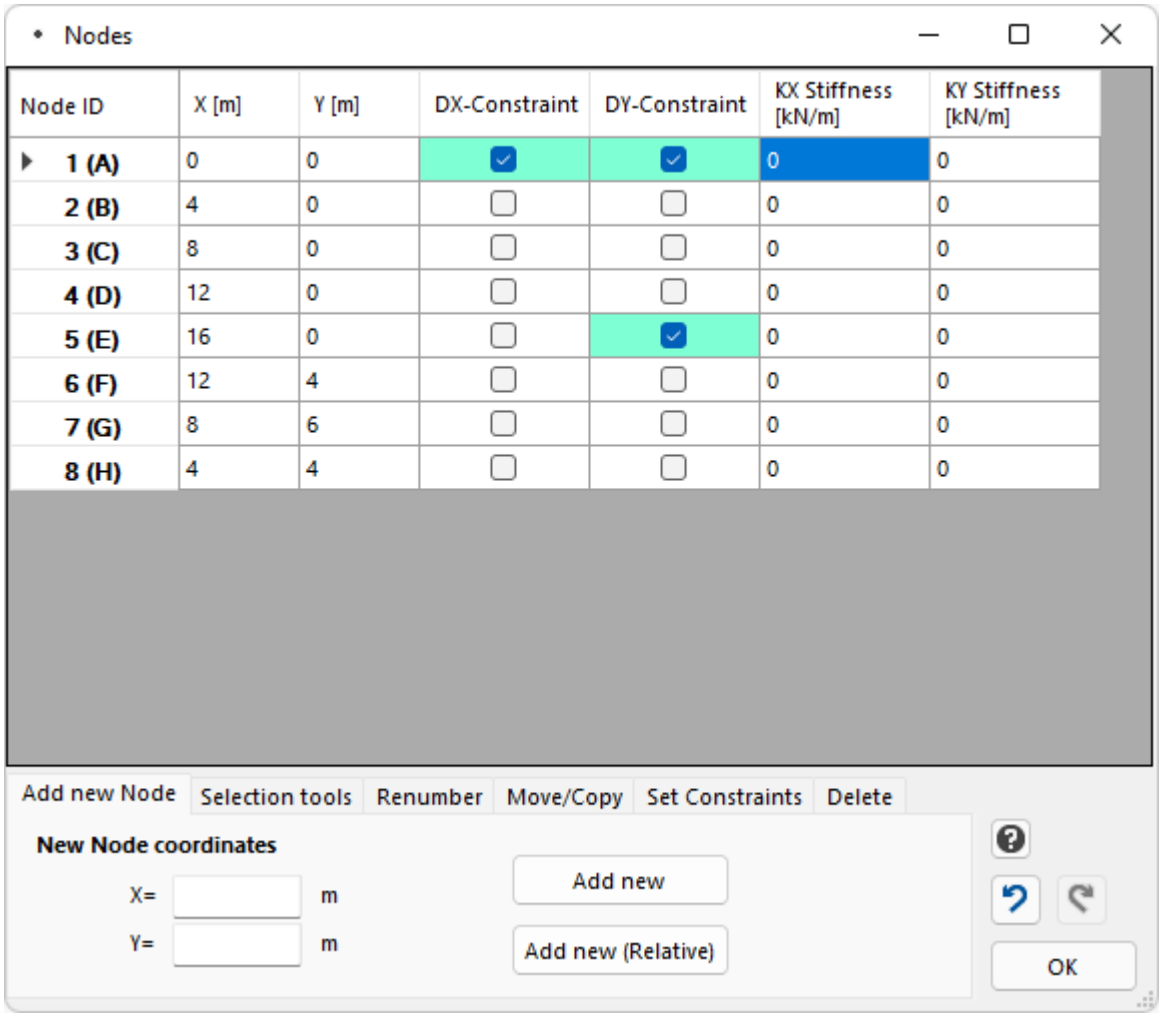
- Move Nodes to their exact positions, if needed.
- **Define or change Nodal Constraints (Supports)**.
- Define Springs, if needed.

Click to edit **Nodes**. For details, see Nodes.



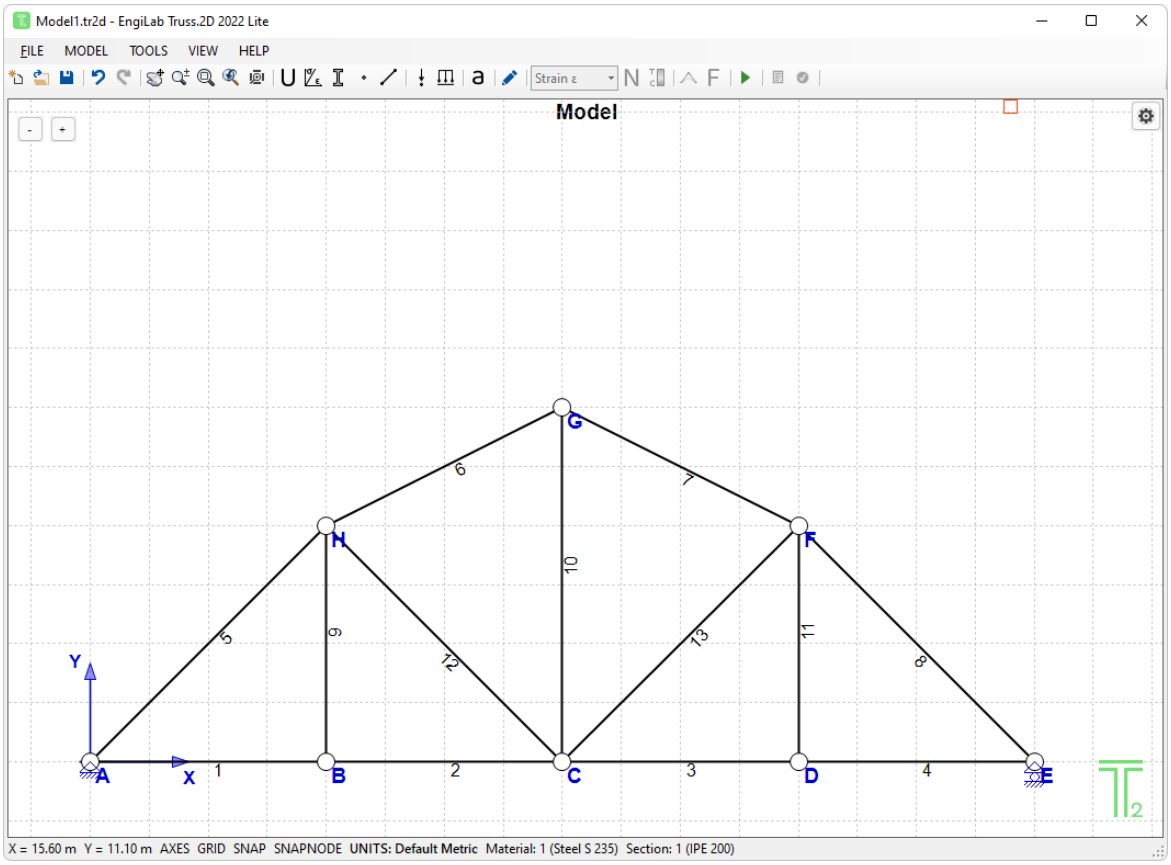
(c) 2022 EngiLab PC

In our example, there is no need to move Nodes, as they are already in their correct positions. What we need to do is to add Constraints (Supports) to the Model. **Set the Constraints for Nodes A and E** as follows.



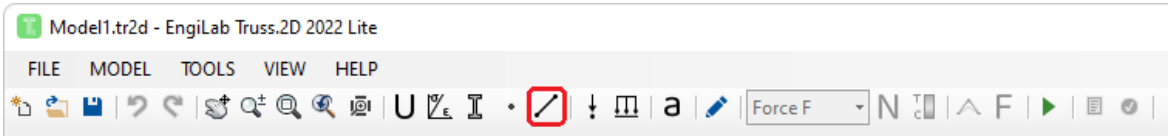
(c) 2022 EngiLab PC

Then **Click OK**. The Model should now look like this:



The supports have been added to Nodes A and E as shown above.

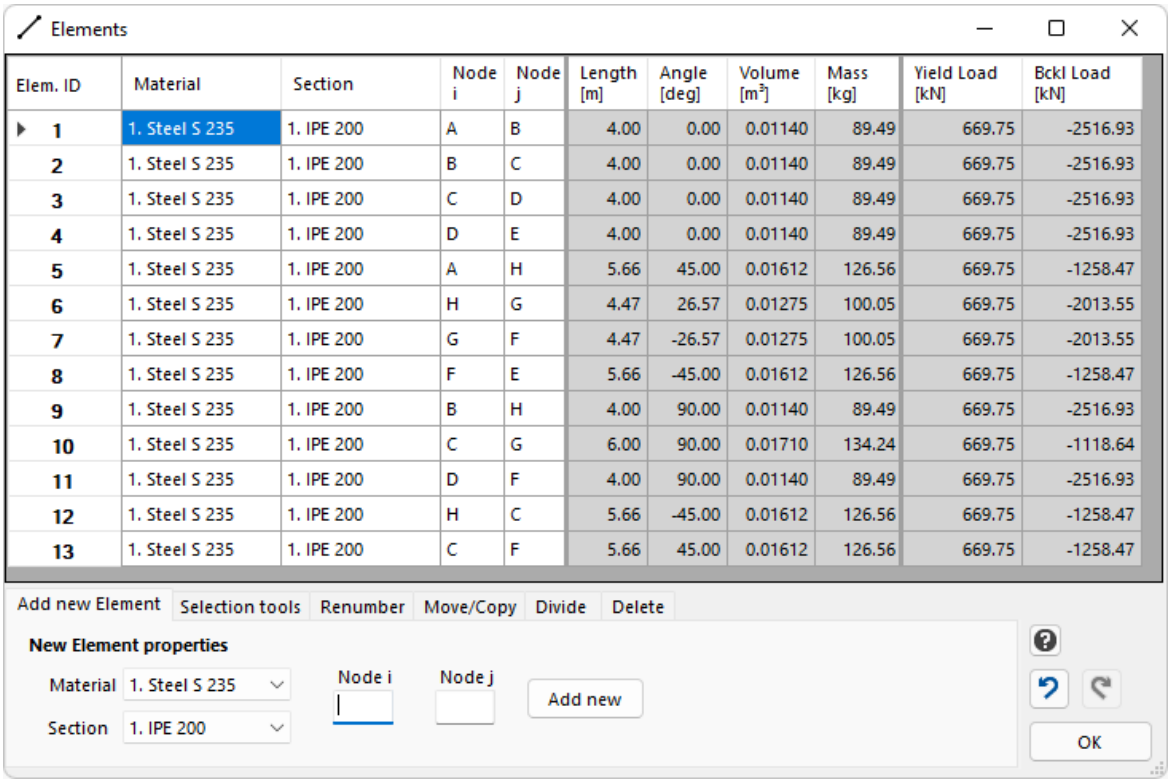
6.1.7 Step 6. Edit Elements



Now let's see the Elements. In our example, the Elements are correct and there is no need to change anything. In other cases, you may need to change the Material or the Section of some Elements, in case more than one Material or Section groups have been defined and some Elements should not belong to the Default Material or Section. You can also change the connectivity of the Elements, i.e. the Start (i) and End (j) Node of any Element.

(c) 2022 EngiLab PC

Click to edit **Elements**. For details, see Elements.



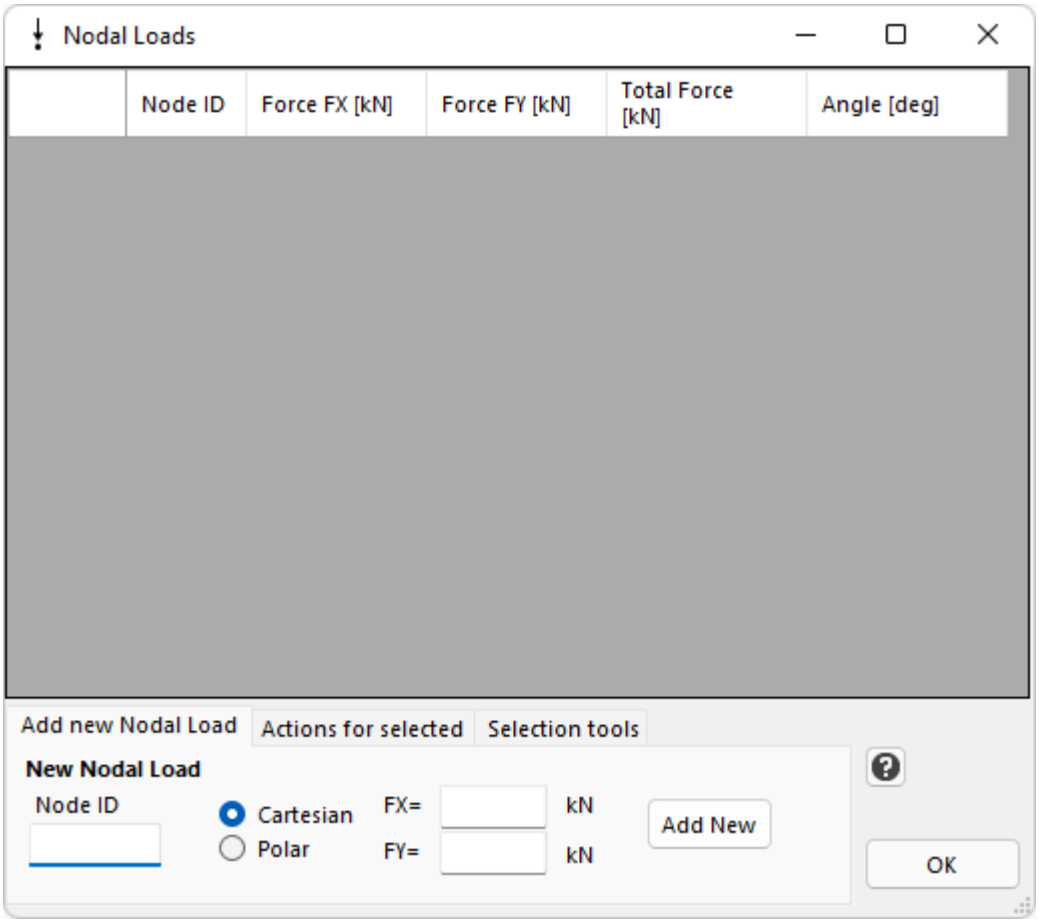
Then **Click OK** to exit Elements.

6.1.8 Step 7. Define Nodal Loads



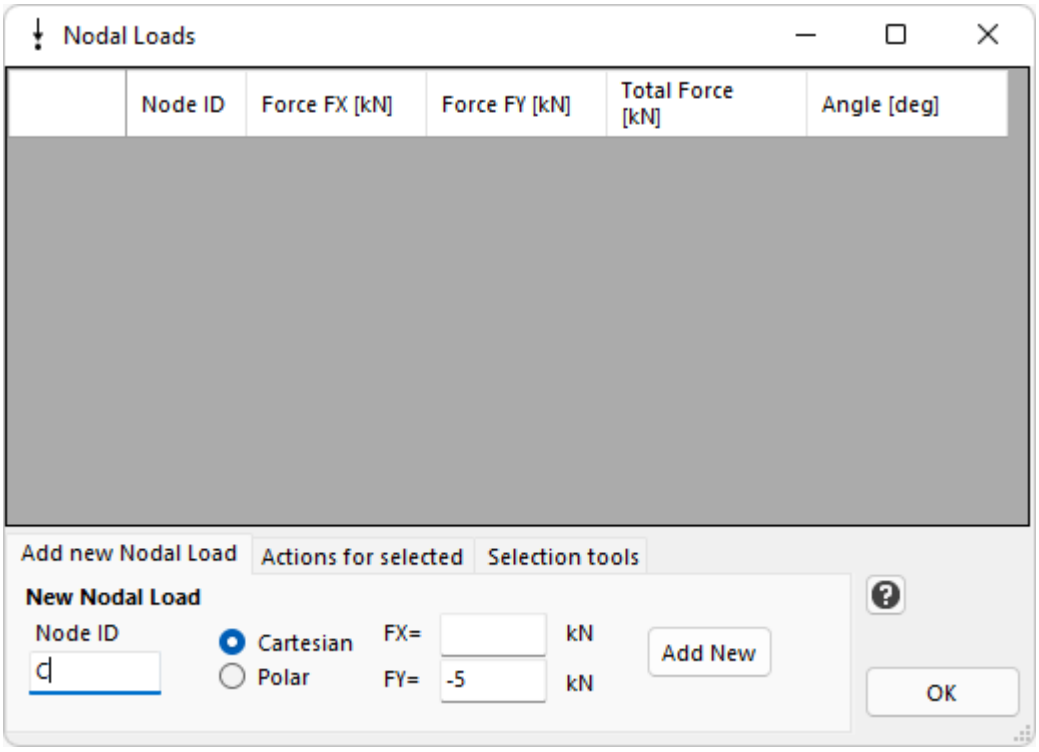
Click to define **Nodal Loads**. For details, see Nodal Loads.

The Nodal Loads form appears.

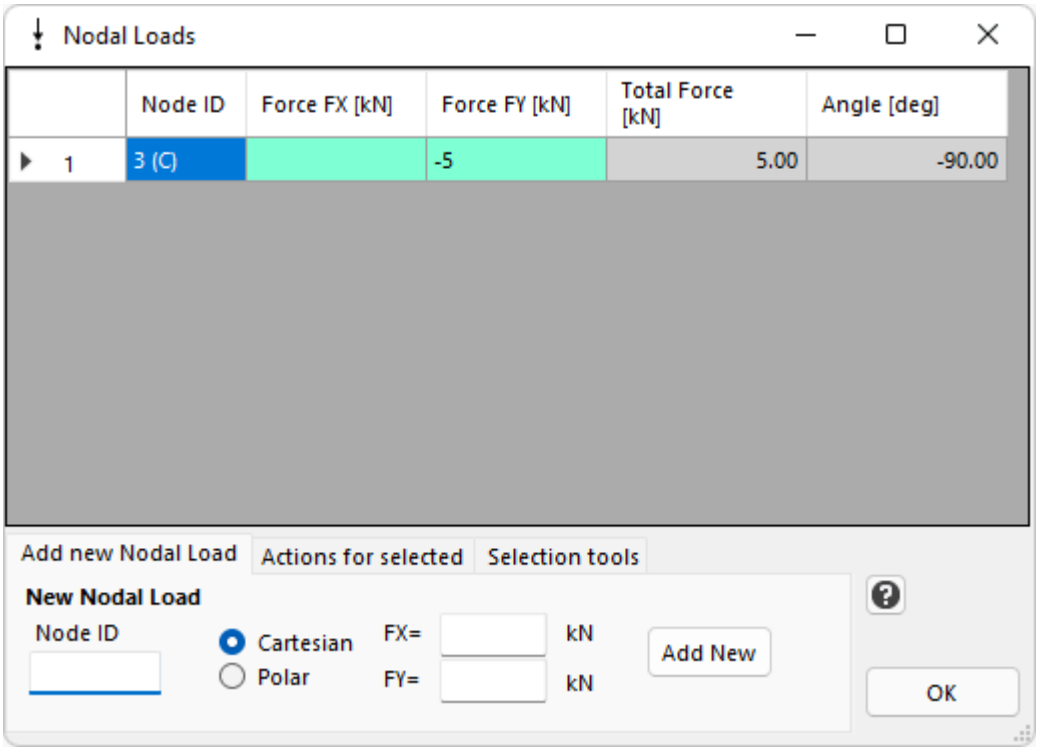


(c) 2022 EngiLab PC

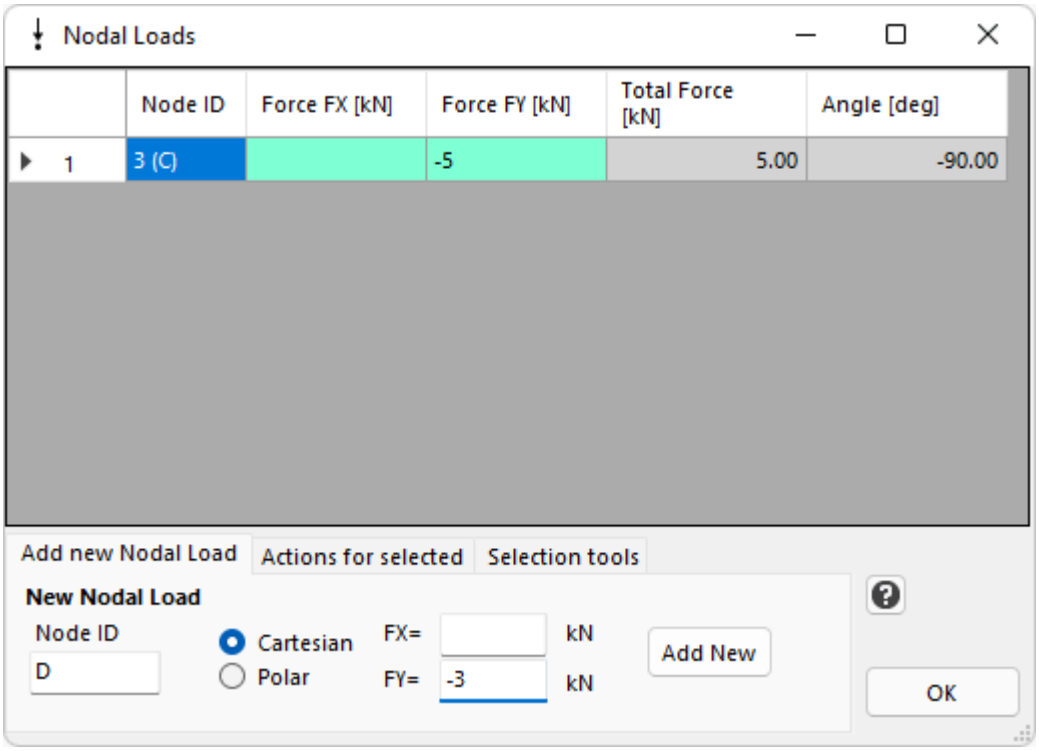
Add a Nodal Load **FY=-5 kN for Node C**.



Click "Add New". You get the following.

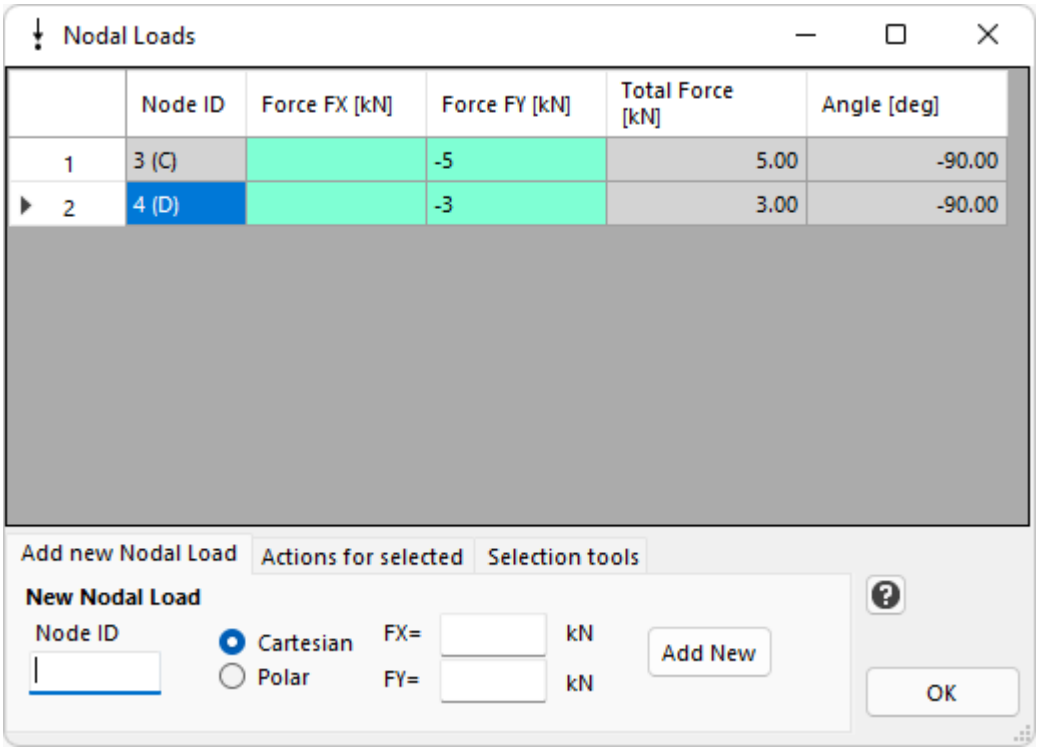


Then add a Nodal Load **FY=-3 kN for Node D**.

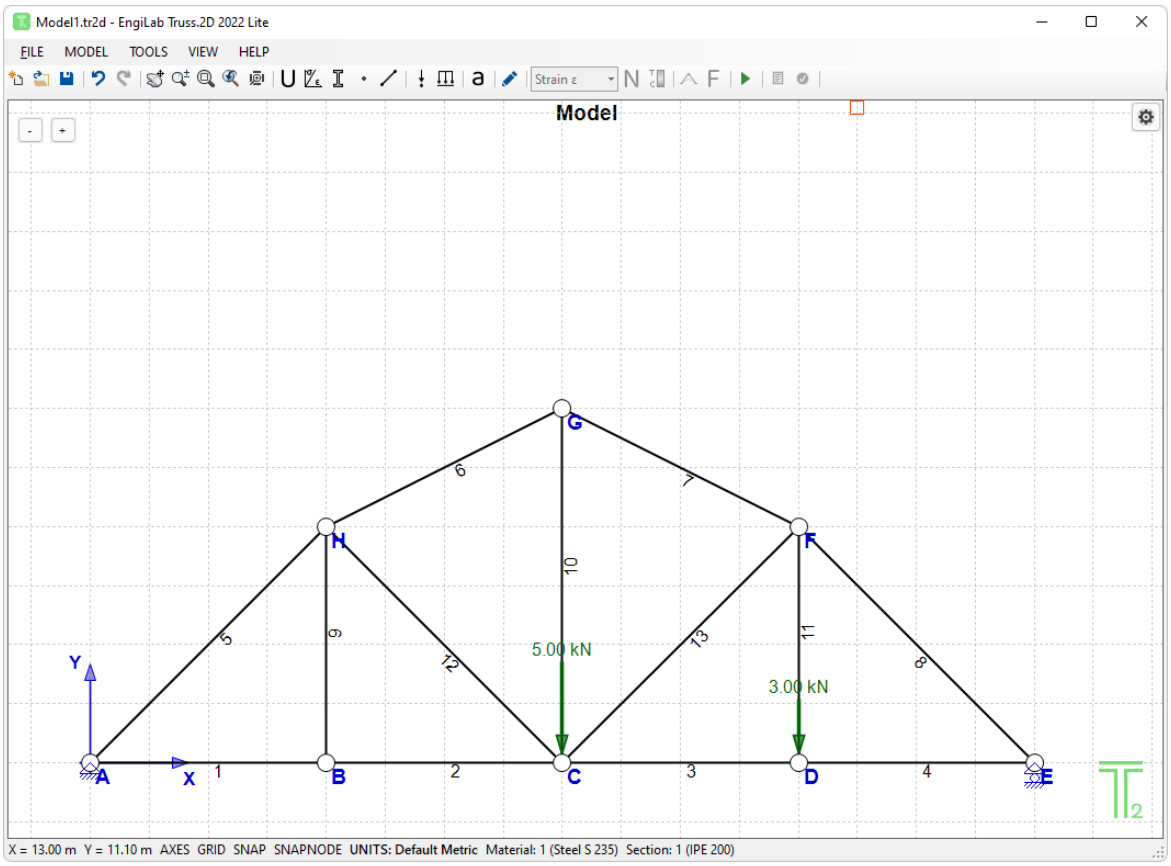


(c) 2022 EngiLab PC

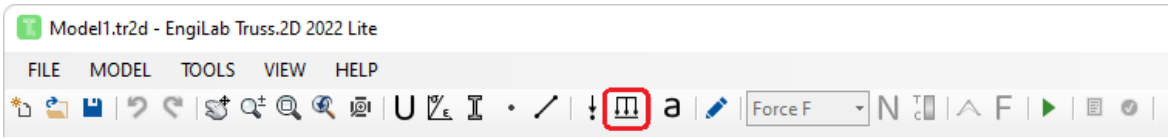
Click "Add Nodal Load". Now the final Nodal Loads should look like this.



Click OK to exit. Now the Model should look like this. The Nodal Loads have been applied!



6.1.9 Step 8. Define Elemental Loads

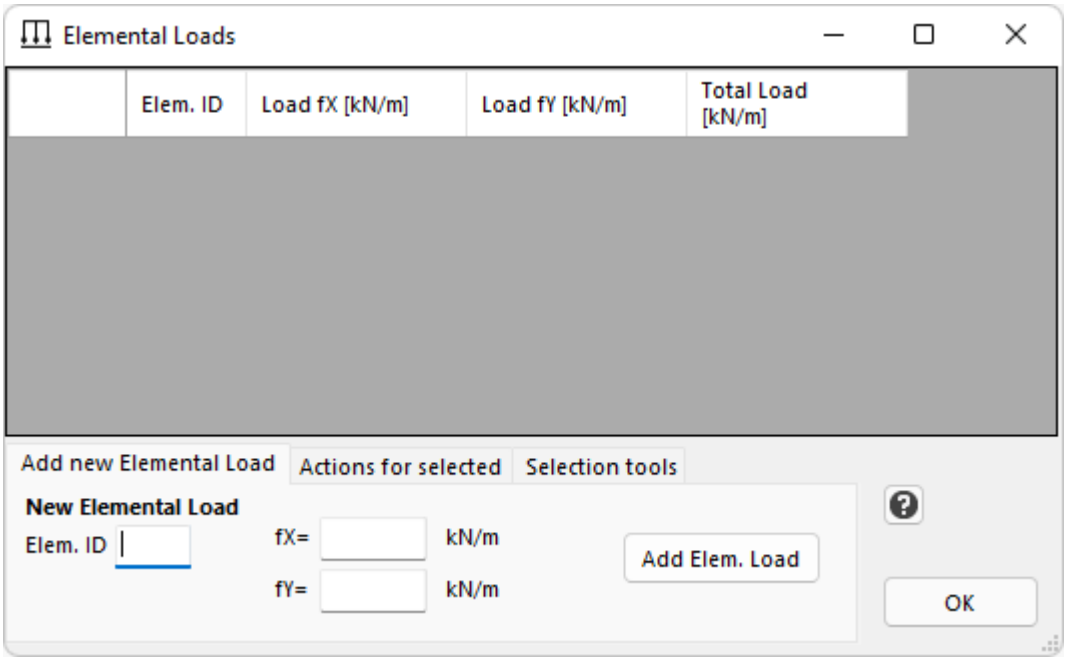


In our example, there are no Elemental Loads, so this step can be skipped.

Click to define **Elemental Loads**, in case there are any. For details, see Elemental Loads.

(c) 2022 EngiLab PC

The Elemental Loads form appears.



Click OK to exit.

6.1.10 Step 9. Define Body (Acceleration) Loads

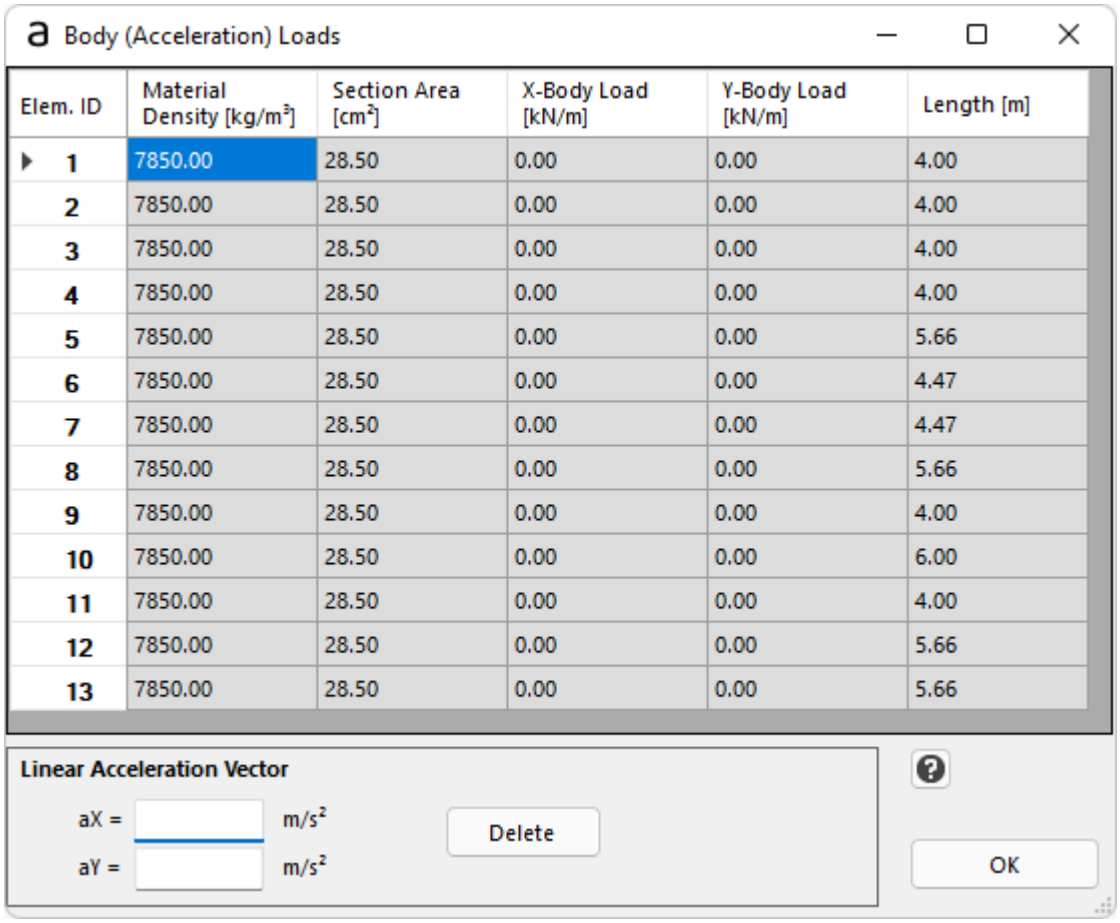


In our example, there are no Body (Acceleration) Loads, so this step can be skipped.

Click a to define **Body (Acceleration) Loads**, in case there are any. For details, see Body (Acceleration) Loads.

(c) 2022 EngiLab PC

The Body (Acceleration) Loads form appears.



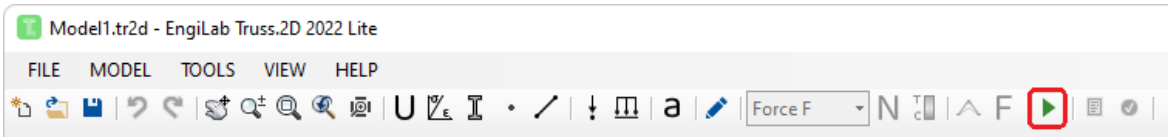
Click OK to exit.

Note: If the want to take into account the self-weight of Elements as an additional elemental load for each Element, we have to provide the **Material Density** for the Material of each Element (we have done that in this example) and also to define a **Linear Acceleration Vector** equal to the **standard earth gravitational acceleration**. We should set the earth gravitational acceleration with a minus (-) sign at the Y direction, i.e. gravity acting towards -Y global axis.

Using the Default Metric System of Units, we need to **input -9.80665, or simply -9.81, at the aY component of the Linear Acceleration Vector** and then press ENTER on the keyboard. The program will automatically calculate the self weight (in kN/m) for each Element, as a Y-Body Load. All properties on the table are read-only properties.

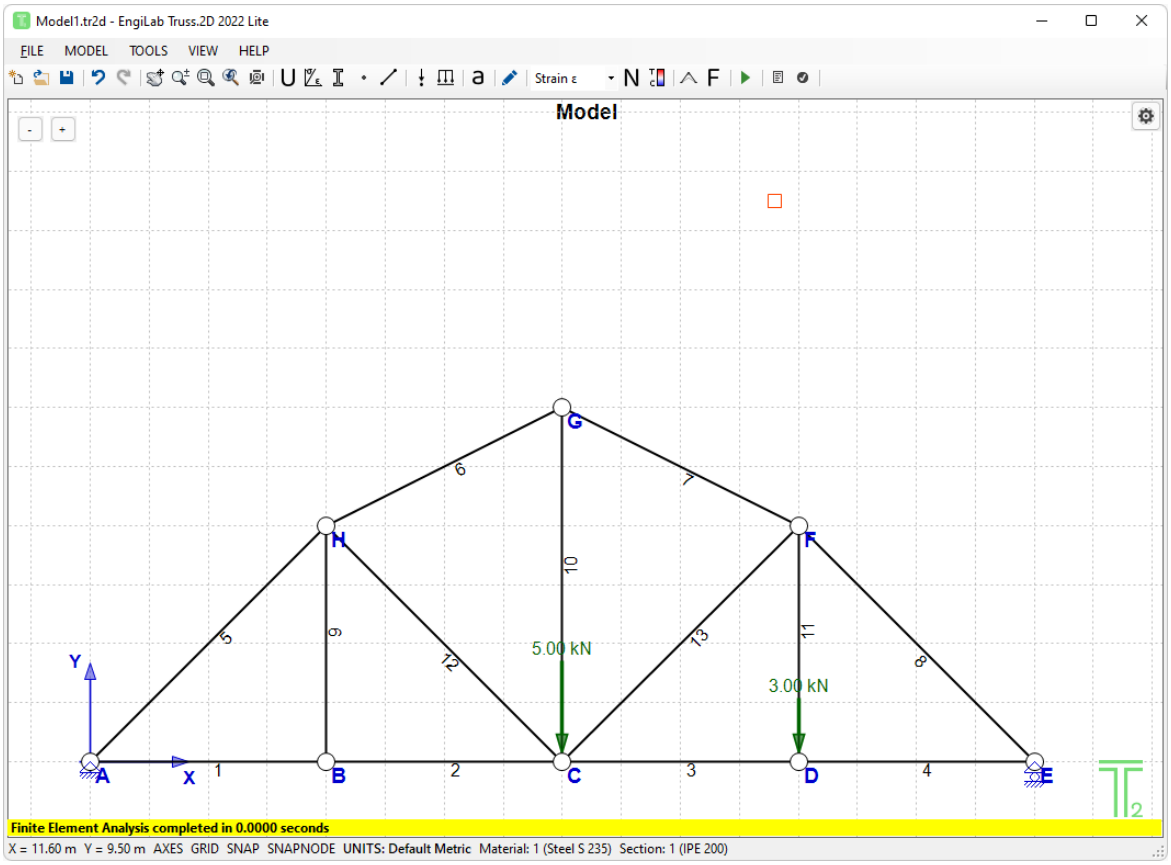
(c) 2022 EngiLab PC

6.1.11 Step 10. Run the Analysis



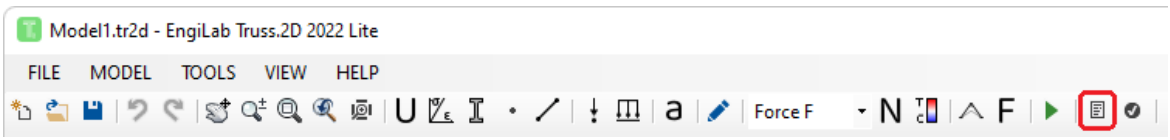
Click (or press F5 on the keyboard) to **run the Finite Element Analysis**. For details on the Analysis, see Analysis.

You should see the following picture.



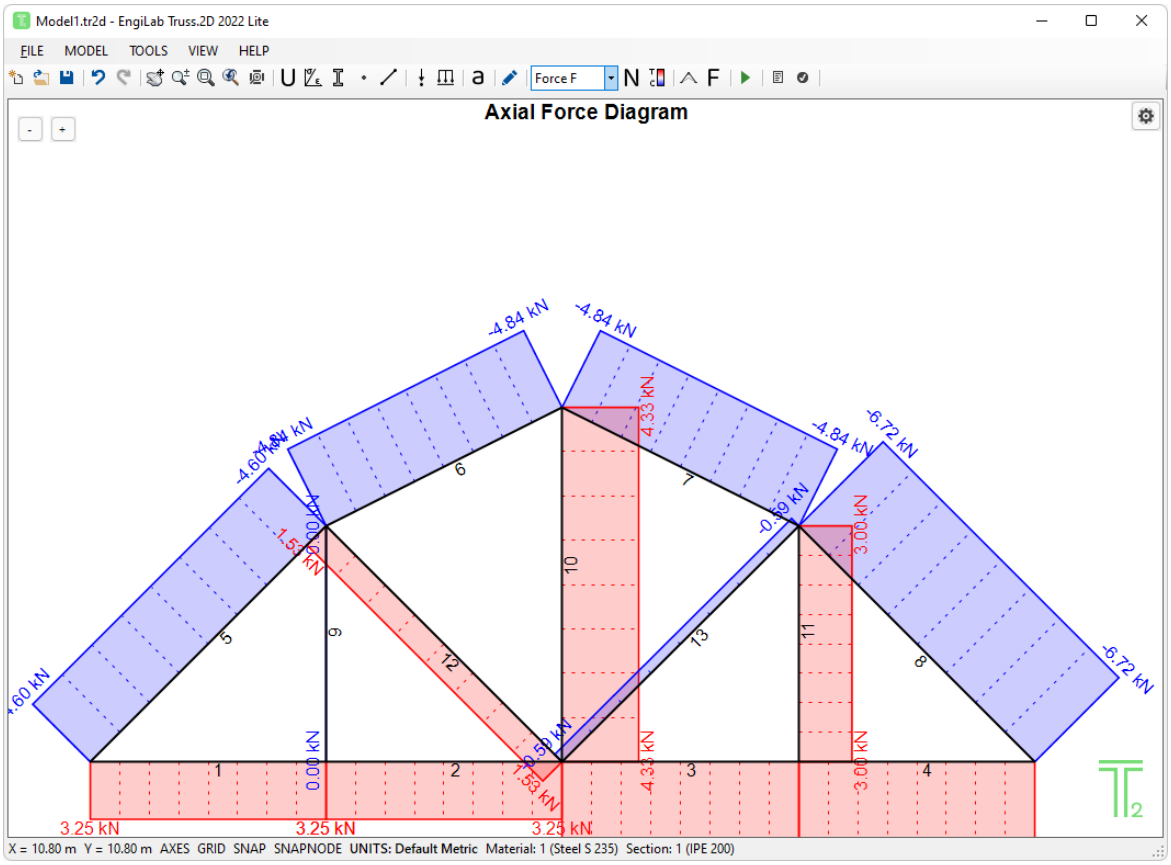
The Finite Element Analysis has been completed successfully.

6.1.12 Step 11. View the Analysis Results on screen



1. Axial Force/Stress/Strain Diagrams

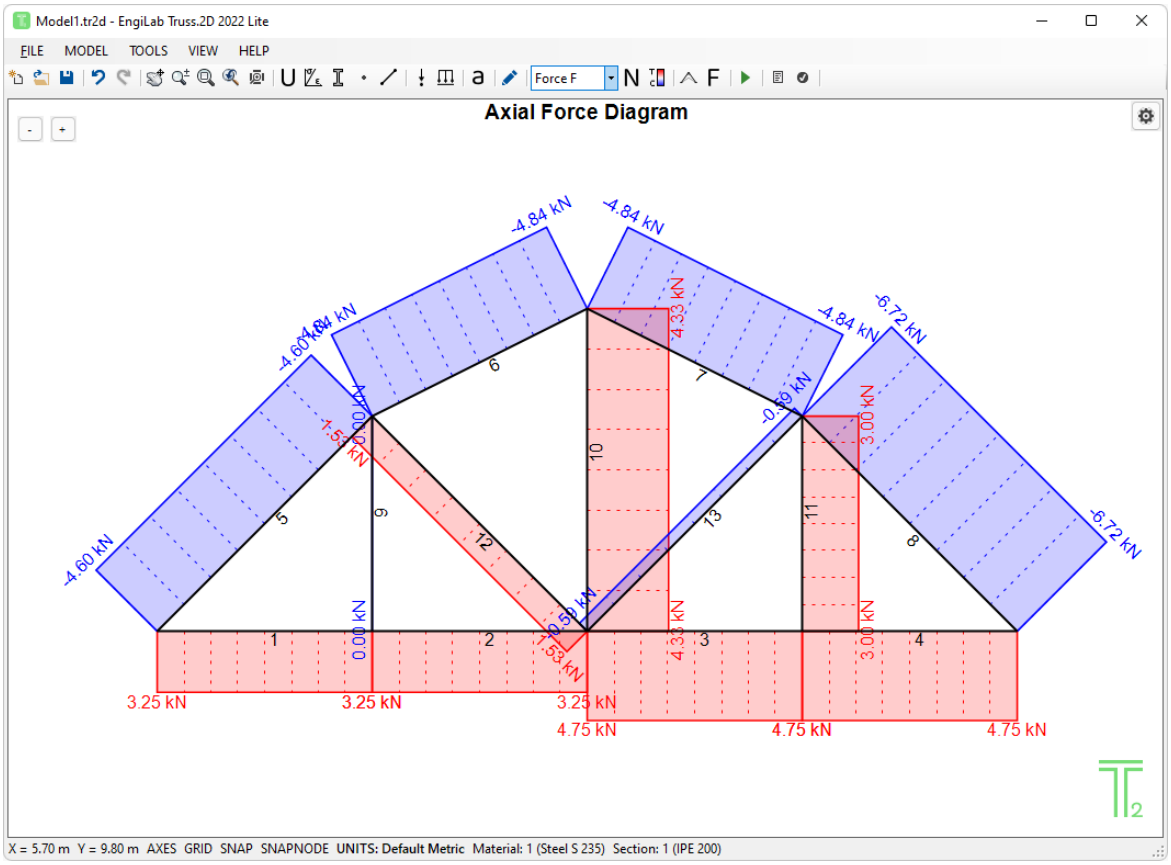
Click N, and select "**Force F**" on the Dropdown menu (on the left of the N button), to see the **Axial Force Diagram**. You may take the picture below.



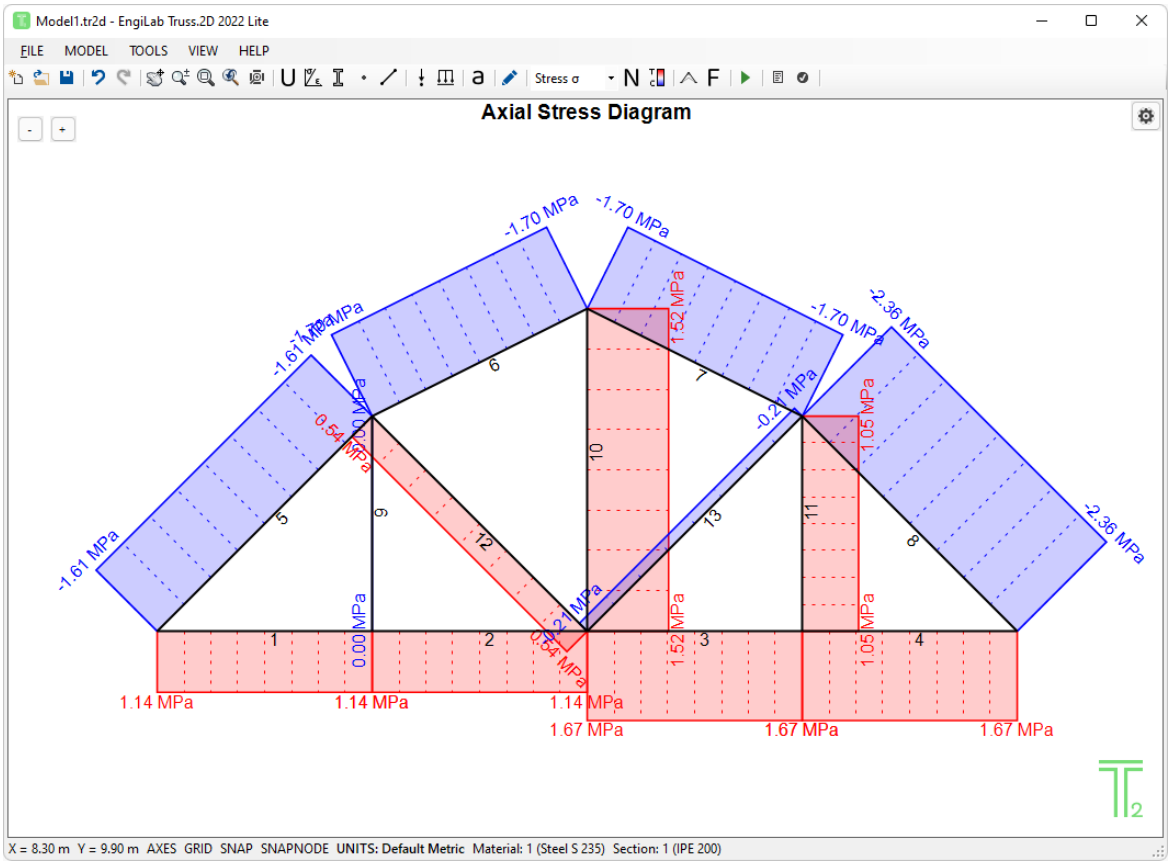
The Axial Force Diagram is out of Scale. **Click the "Zoom All" button** to automatically scale the Diagram.

(c) 2022 EngiLab PC

Now the result should look like this, which is much better. We see that the Force of Element 13 (Member CF) is -0.59 kN, the same as the one we found with Hand Calculations.

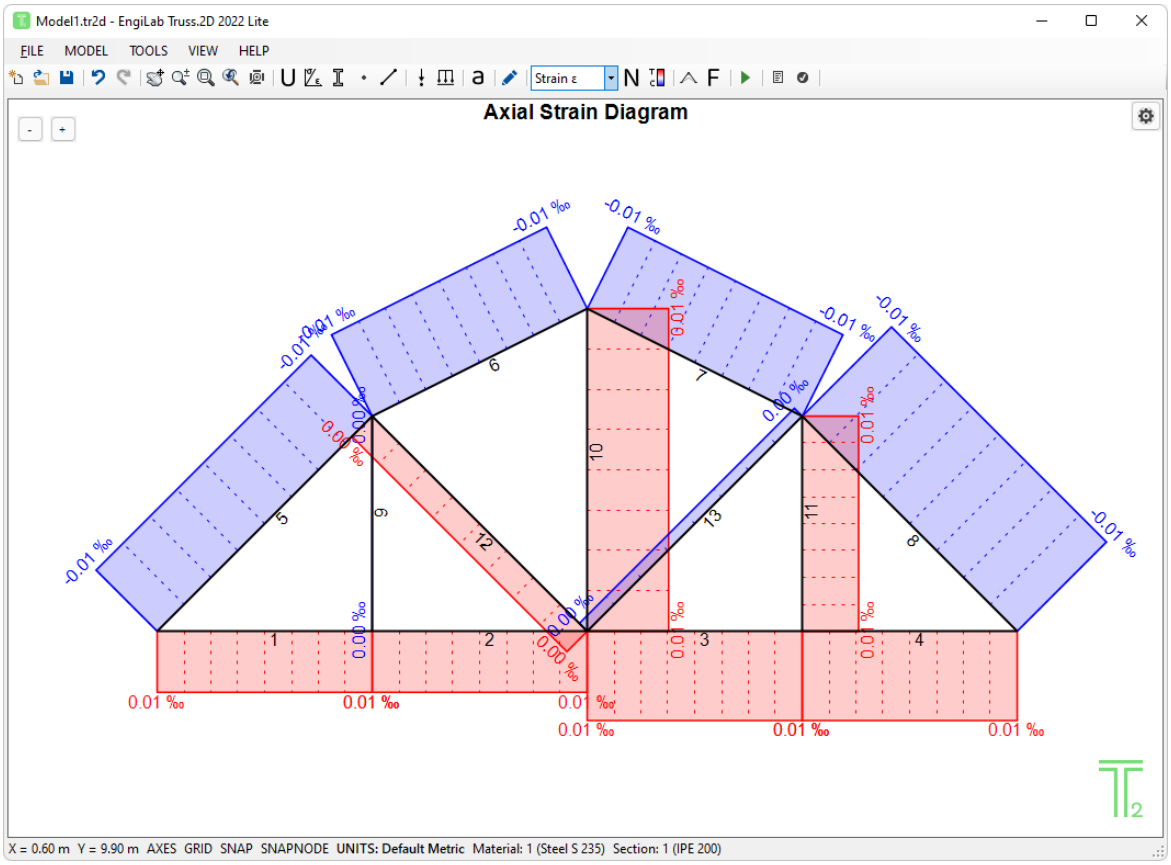


Select "**Stress σ**" on the Dropdown menu and click N to see the **Axial Stress Diagram**. If the Axial Stress Diagram is out of Scale, **Click the "Zoom All" button** to automatically scale the Diagram. You should take the following picture.



Again, select "**Strain ε**" on the Dropdown menu and click N to see the **Axial Strain Diagram**. If the Axial Strain Diagram is out of Scale, **Click the "Zoom All" button** to automatically scale the Diagram. You should take the following picture.

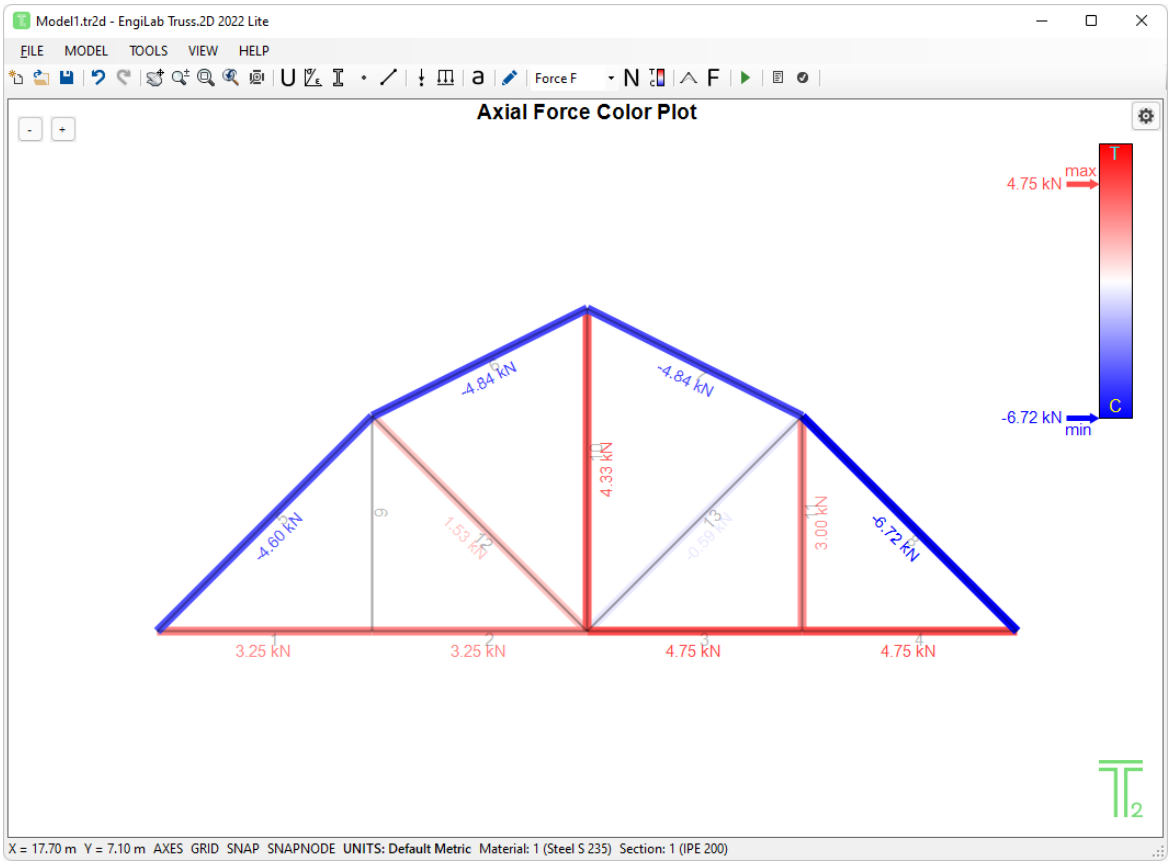
(c) 2022 EngiLab PC



Although the strain is in Permille units (‰), it appears that 2 decimal places are not enough to express it the right way. You may want to increase the number of decimals. To do that, visit the Units form and set the Decimals for strain from 2 to 4. Then you will get more accurate strain values. The Strain Diagram is plotted correctly and is not affected by the number of decimals used. Only the shown values are affected.

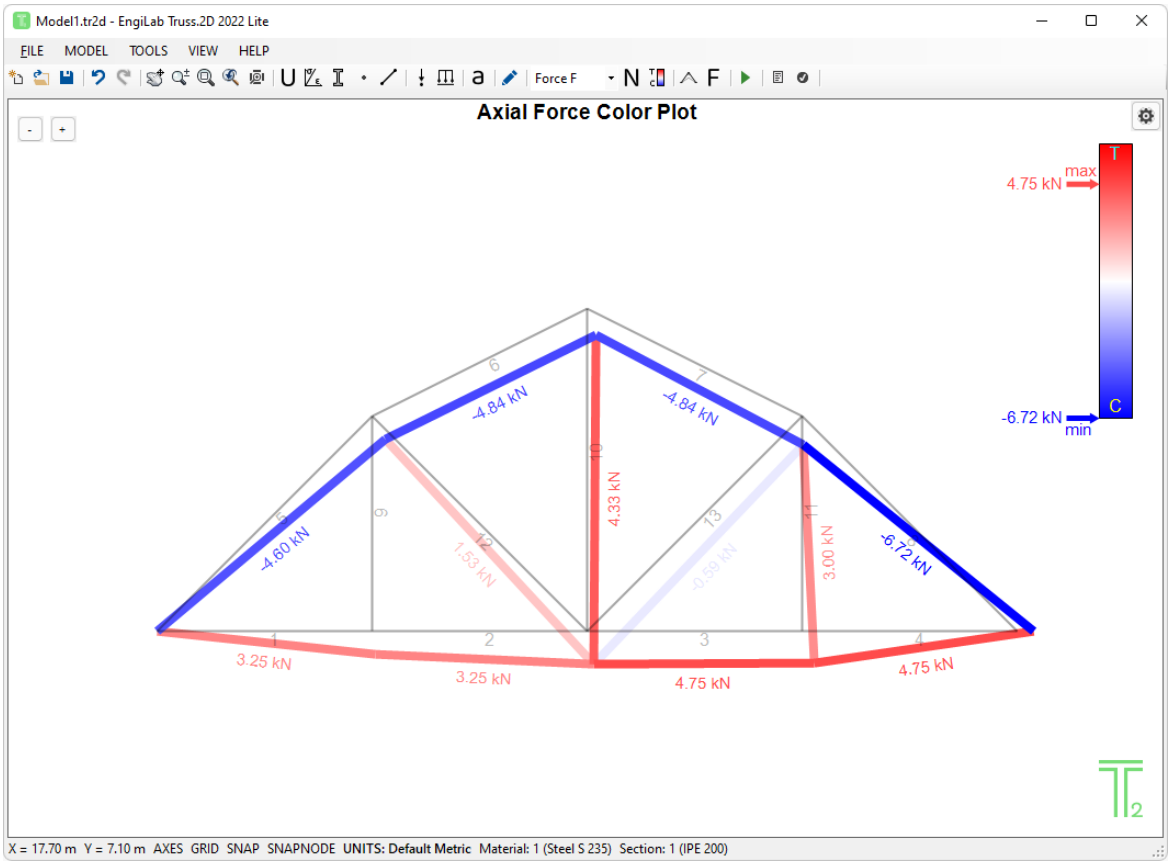
2. Axial Force/Stress/Strain Color Plots

Click , and select "**Force F**" on the Dropdown menu (on the left of the N button), to see the **Axial Force Color Plot**. You may take the picture below.

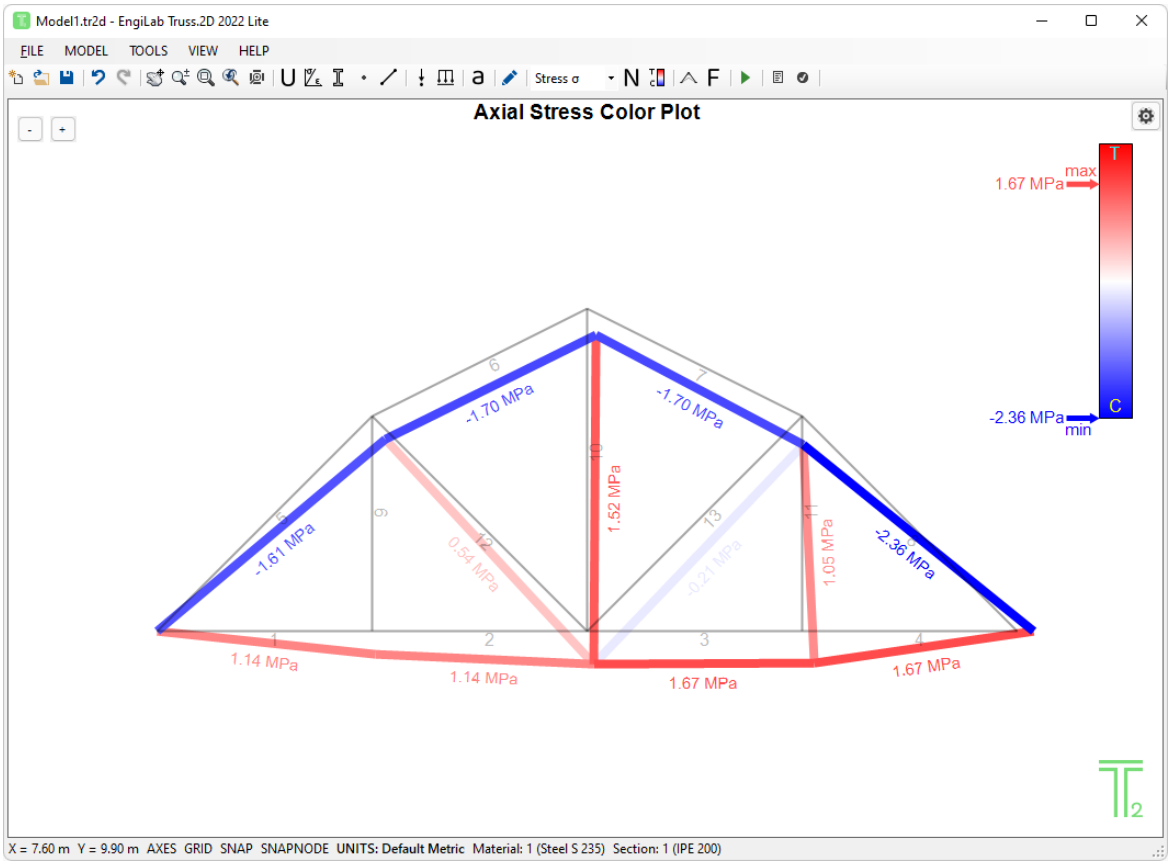


Click the "Zoom All" button to automatically scale the Diagram. Now the result should look like this. Note that the Color Plot is drawn on the Deformed state of the Model, by default. This is a setting which can be changed in Settings.

(c) 2022 EngiLab PC

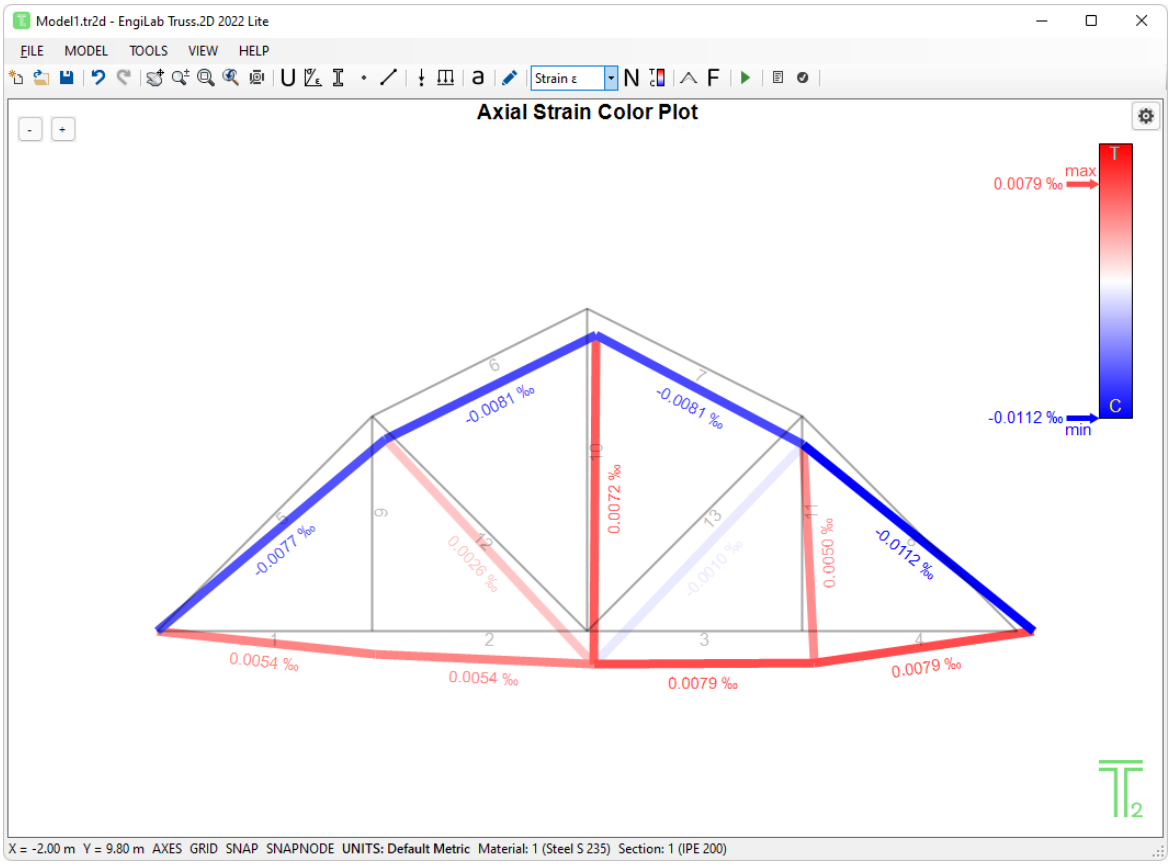


Select "**Stress σ**" on the Dropdown menu and click to see the **Axial Stress Color Plot**. If deformation shown on the Axial Stress Color Plot is out of Scale, **Click the "Zoom All" button** to automatically scale the Diagram. You should take the following picture.



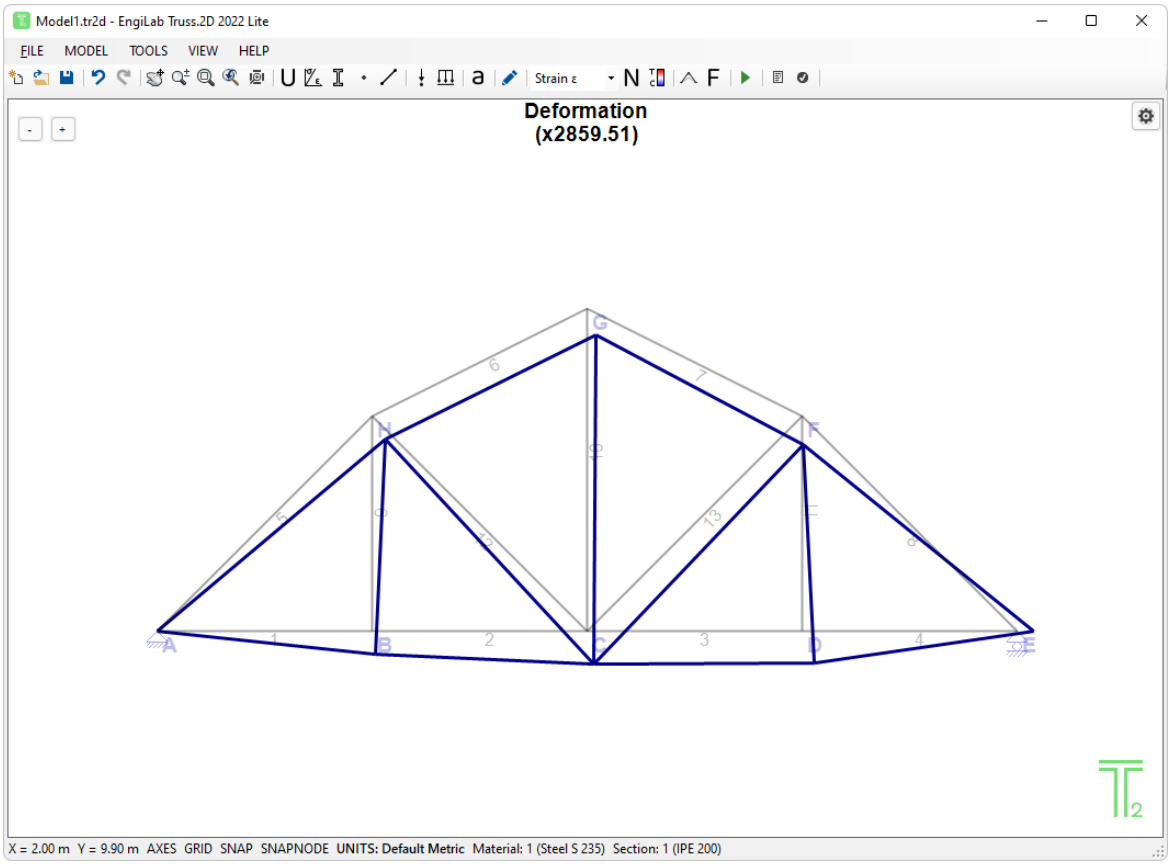
Again, select "**Strain ε**" on the Dropdown menu and click to see the **Axial Strain Color Plot**. **Click the "Zoom All" button** to automatically scale the deformation on the Diagram, if needed. You should take the following picture. Note that we have changed the Decimals from 2 to 4 to depict strain values better.

(c) 2022 EngiLab PC


Model1.tr2d - EngiLab Truss.2D 2022 Lite
FILE MODEL TOOLS VIEW HELP
Strain ε
Axial Strain Color Plot
T
max
0.0079 ‰
-0.0112 ‰
min
C
-0.0081 ‰
-0.0081 ‰
-0.0077 ‰
0.0026 ‰
0.0072 ‰
0.0010 ‰
0.0050 ‰
-0.0112 ‰
0.0054 ‰
0.0054 ‰
0.0079 ‰
0.0079 ‰
X = -2.00 m Y = 9.80 m AXES GRID SNAP SNAPNODE UNITS: Default Metric Material: 1 (Steel S 235) Section: 1 (IPE 200)

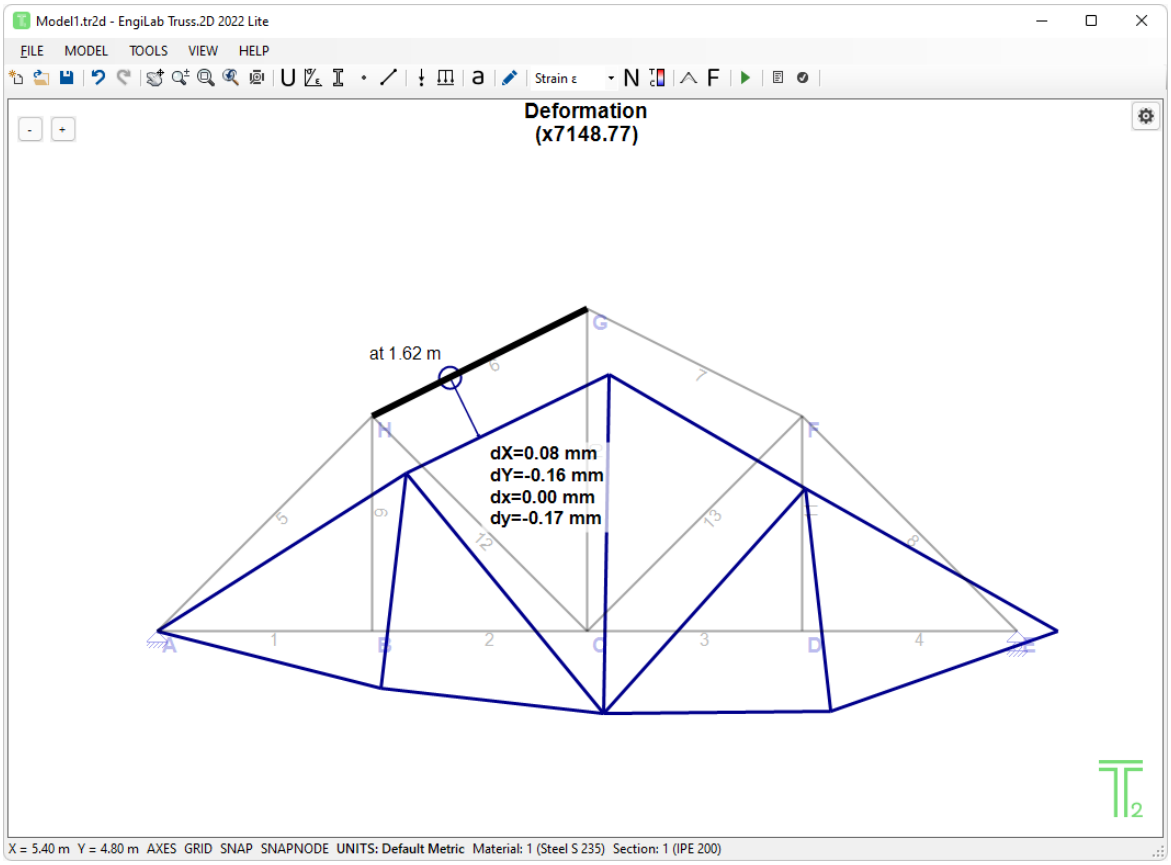
3. Deformation

Click , to see the Model Deformation. If the Deformation is out of Scale, **Click the "Zoom All" button** to automatically scale it. You should take the following picture. The program reports also the Deformation magnification, in our example x2859.51.



(c) 2022 EngiLab PC

You can adjust scaling yourself by using the +/- buttons at the top left of the picture. If you click the "+" button a few times, you may get a picture like the one below where the magnification factor is now x7148.77.



See above that you can get the deformation values on screen, if the mouse pointer hovers over an element.

IMPORTANT: The values that are given on screen are the **dX**, **dY**, **dx** and **dy** displacements of the corresponding point of each element.

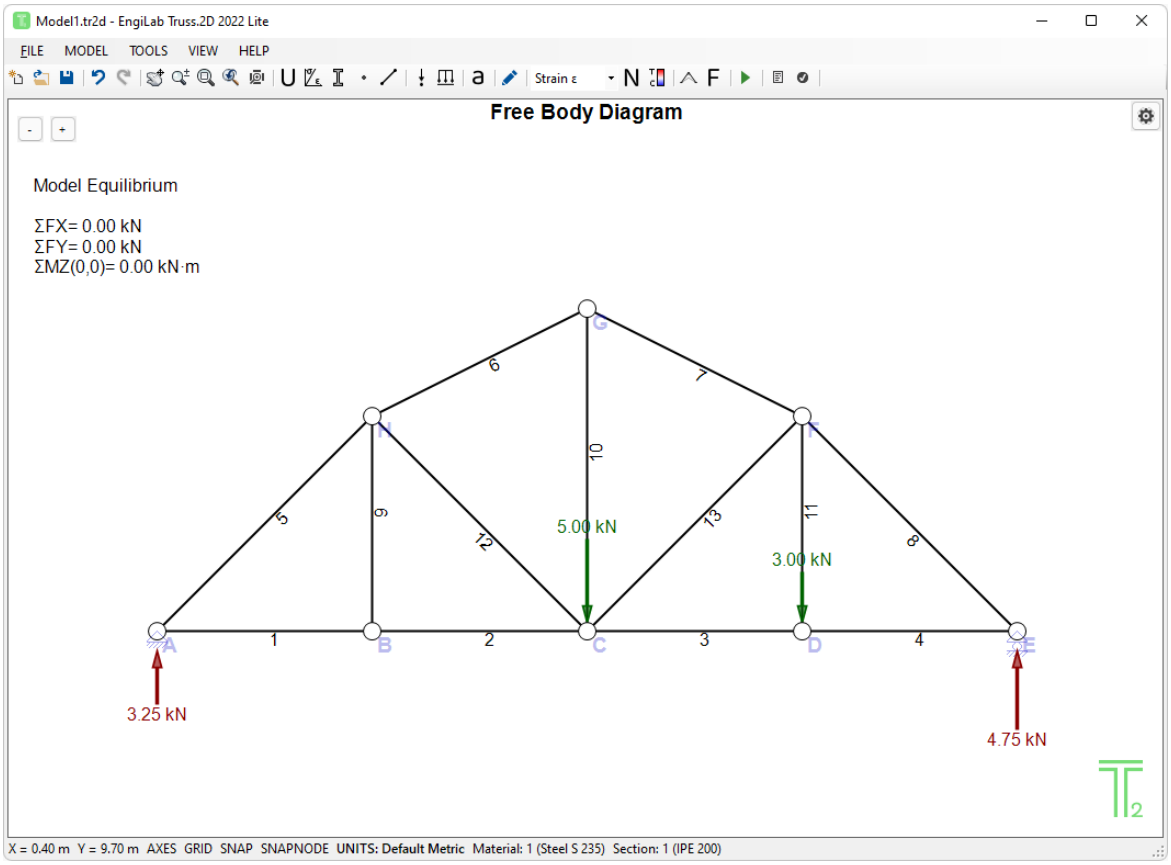
- **dX** and **dY** are the X- and Y- displacements of a point, respectively, in **Global axes**. For example a positive dX means displacement to the right, while a positive dY means displacement upwards.
- **dx** and **dy** are the x- and y- displacements of a point, respectively, in **Local Element Axes**, where x is defined from the Start Node i and the End Node j of the Element and y is perpendicular to it.

(c) 2022 EngiLab PC

4. Free Body Diagram (FBD)

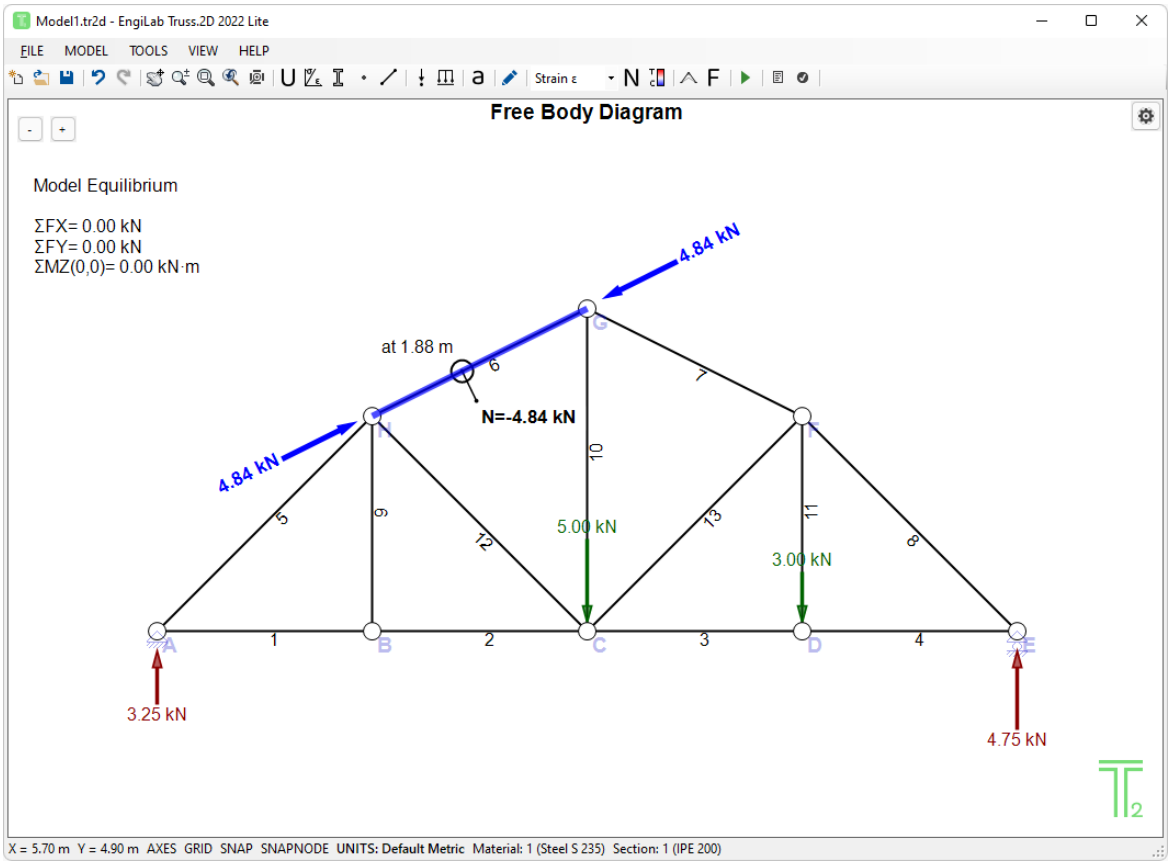
Click F, to see the Free Body Diagram of the Model. The Free Body Diagram shows the support reactions on screen and also the calculations of the equilibrium of the Model.

We see that the support reactions are the same as the ones found with Hand Calculations.

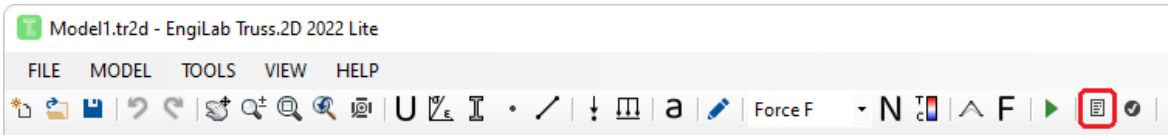


(c) 2022 EngiLab PC

Note that if you hover the pointer over an Element, you can read the corresponding Axial Force at that point, as shown below. The Element End Forces are also given on screen, for the specific Element.

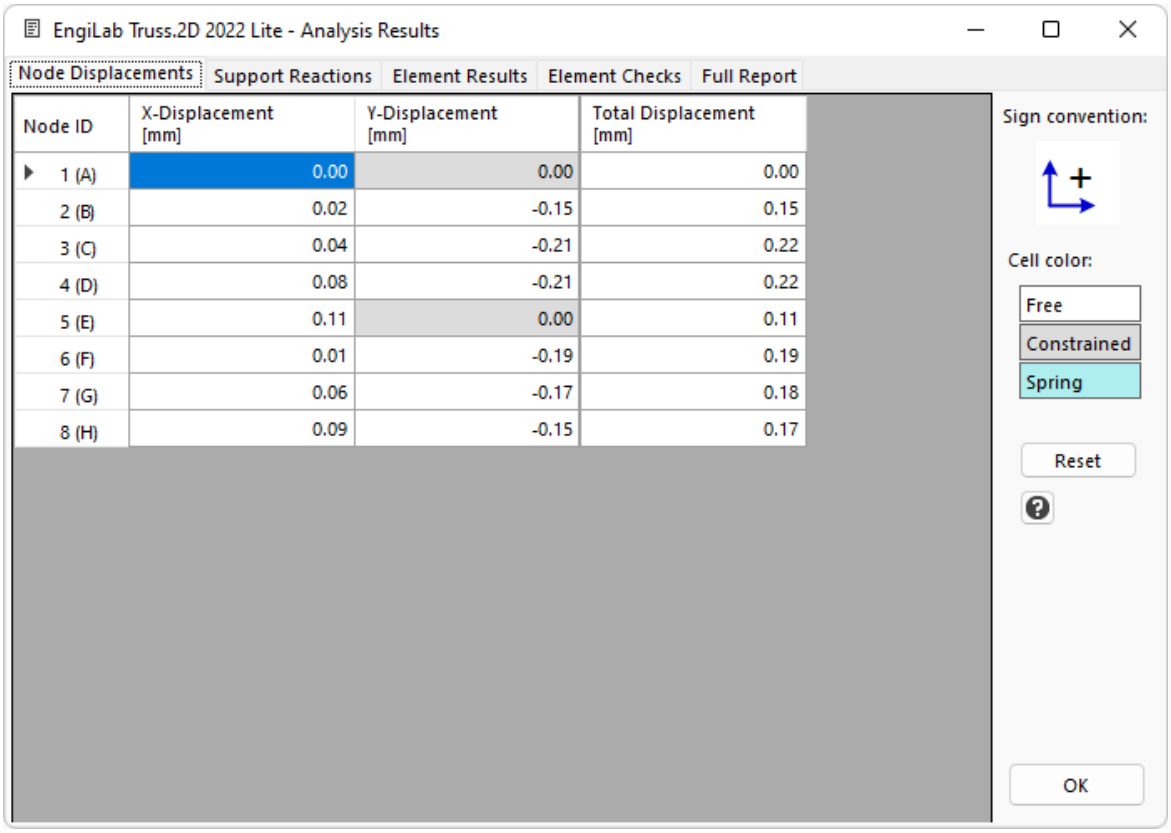


6.1.13 Step 12. View the analytical results



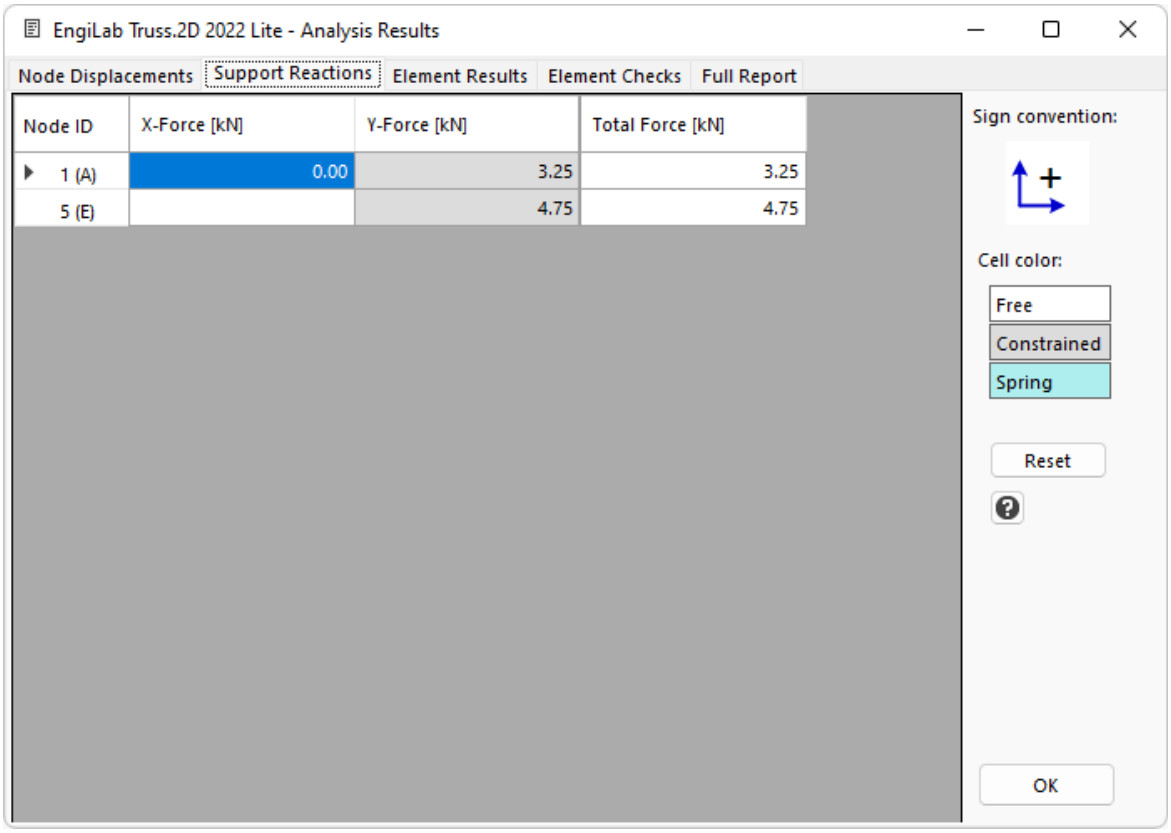
Click to see the analytical results.

1. Node Displacements

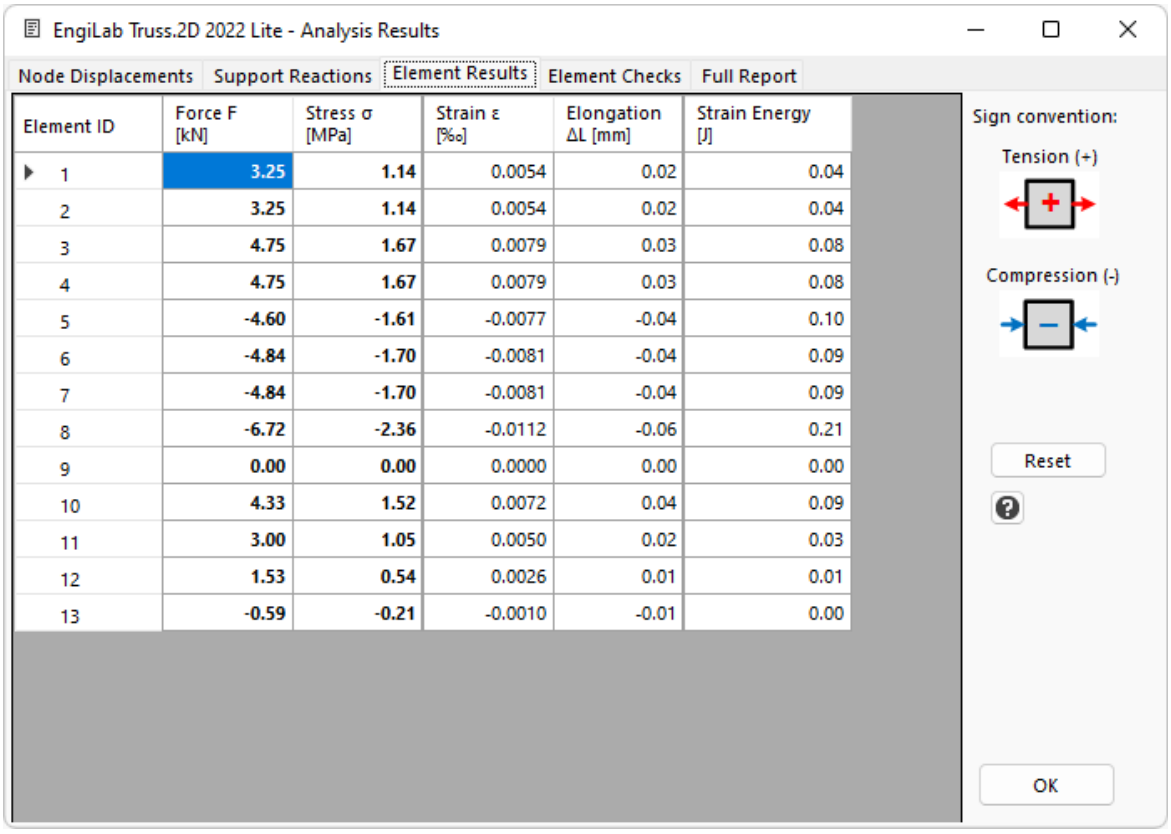


(c) 2022 EngiLab PC

2. Support Reactions



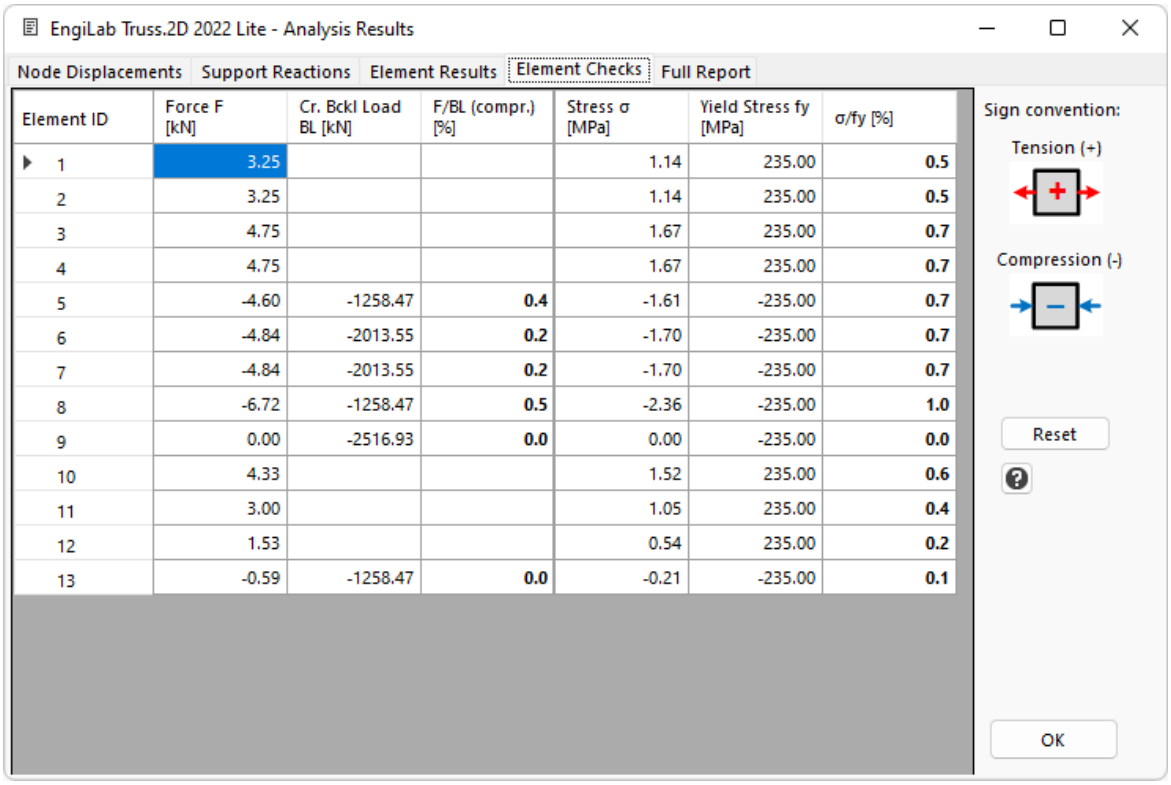
3. Element Results



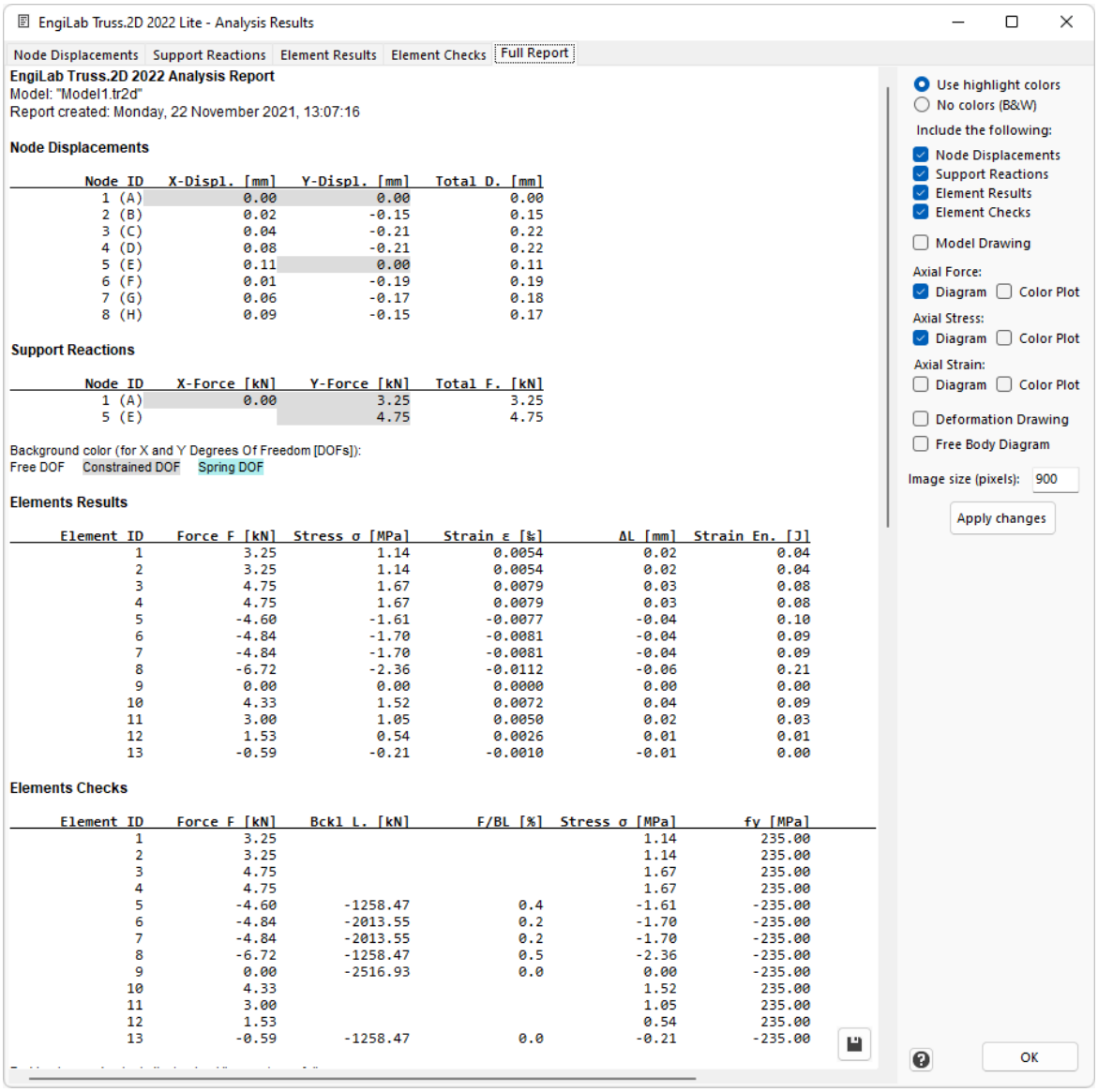
Element ID	Force F [kN]	Stress σ [MPa]	Strain ε [‰]	Elongation ΔL [mm]	Strain Energy [J]
1	**3.25**	**1.14**	0.0054	0.02	0.04
2	**3.25**	**1.14**	0.0054	0.02	0.04
3	**4.75**	**1.67**	0.0079	0.03	0.08
4	**4.75**	**1.67**	0.0079	0.03	0.08
5	**-4.60**	**-1.61**	-0.0077	-0.04	0.10
6	**-4.84**	**-1.70**	-0.0081	-0.04	0.09
7	**-4.84**	**-1.70**	-0.0081	-0.04	0.09
8	**-6.72**	**-2.36**	-0.0112	-0.06	0.21
9	**0.00**	**0.00**	0.0000	0.00	0.00
10	**4.33**	**1.52**	0.0072	0.04	0.09
11	**3.00**	**1.05**	0.0050	0.02	0.03
12	**1.53**	**0.54**	0.0026	0.01	0.01
13	**-0.59**	**-0.21**	-0.0010	-0.01	0.00

(c) 2022 EngiLab PC

4. Element Checks

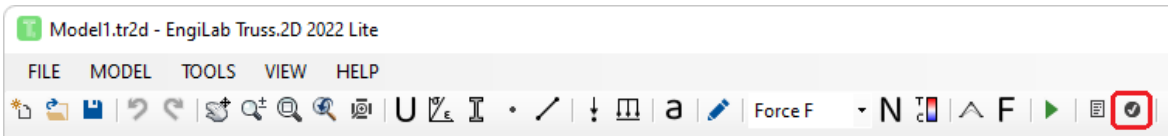


5. Full Report



The Full Analysis Reports may include also the images of the Model, Diagrams, Color Plots, Deformation and Free Body Diagram. You can select the desired items from the Settings on the right of the Report and then click the "Apply changes" button.

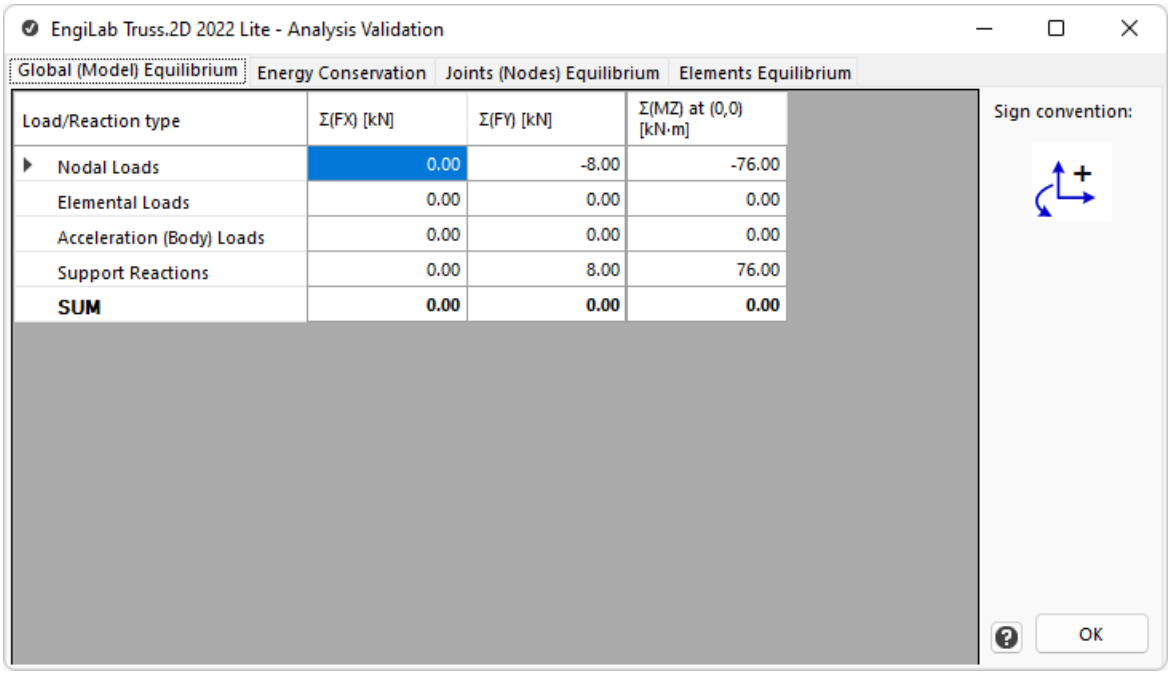
6.1.14 Analysis Validation (optional)



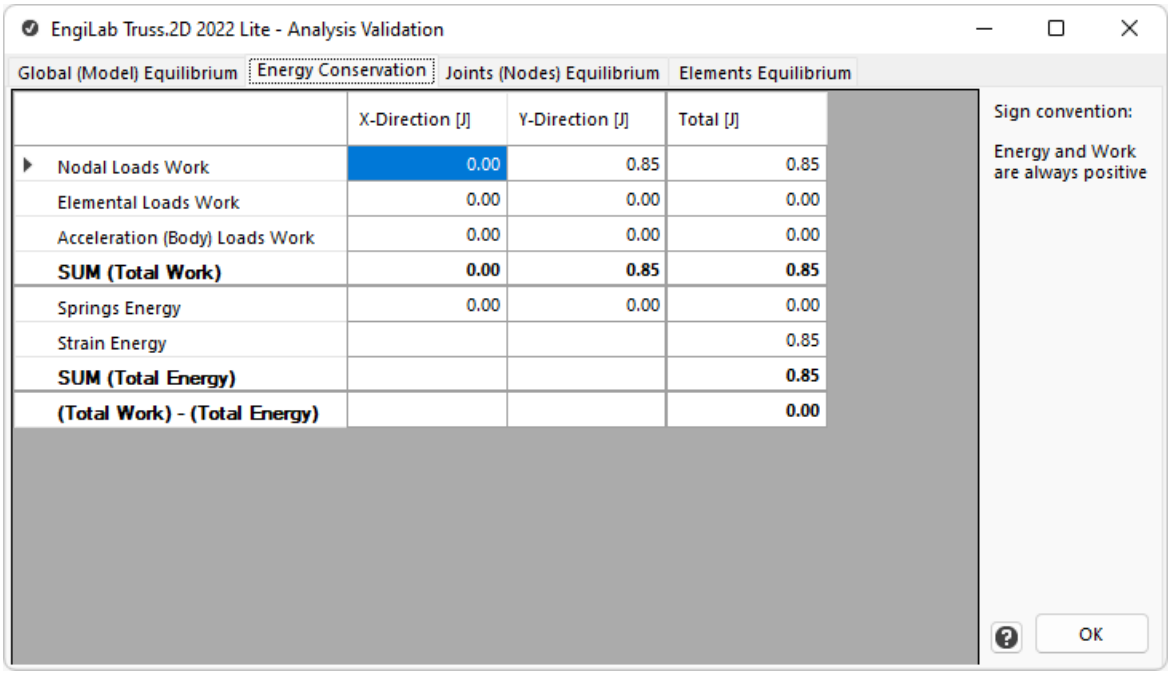
Optionally, you can click to see the Analysis Validation Reports and make sure that everything is OK with the Analysis and its results.

(c) 2022 EngiLab PC

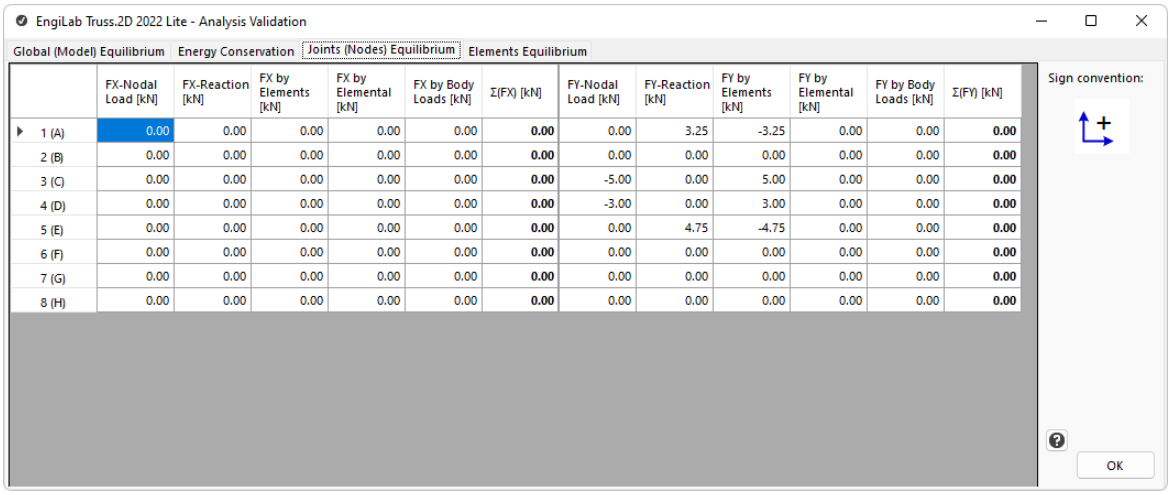
1. Global (Model) Equilibrium



2. Energy Conservation

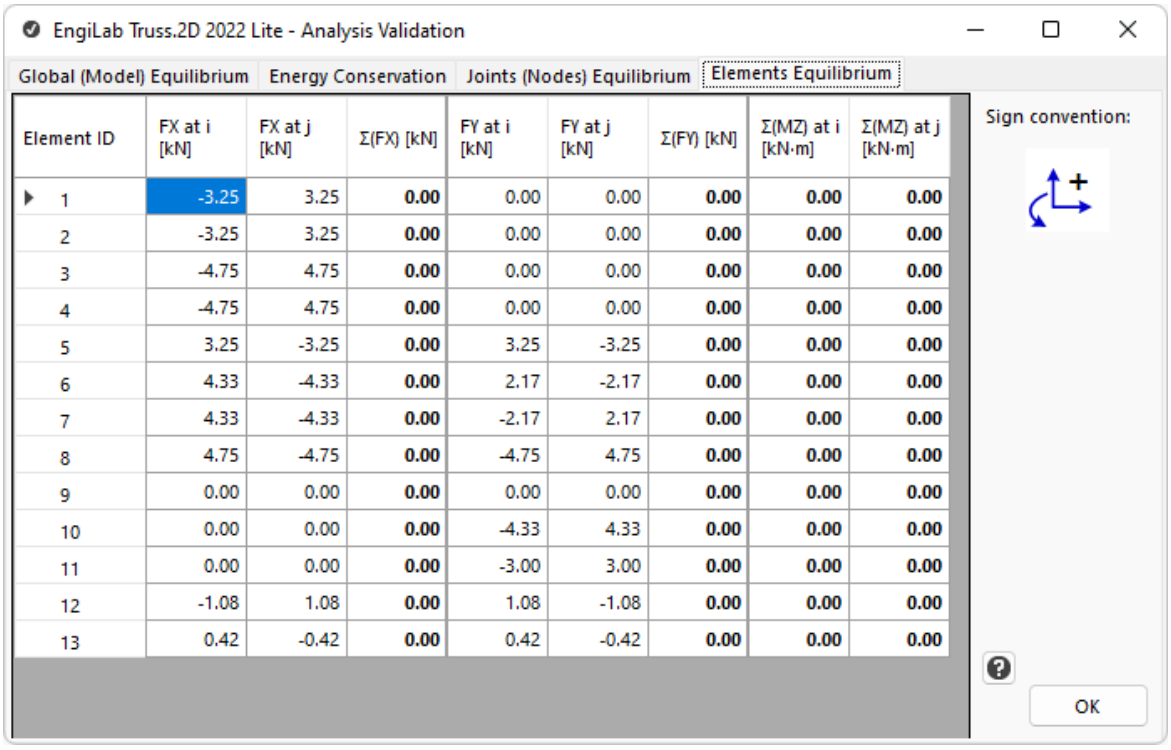


3. Joints (Nodes) Equilibrium



	FX-Nodal Load [kN]	FX-Reaction [kN]	FX by Elements [kN]	FX by Elemental [kN]	FX by Body Loads [kN]	Σ(FX) [kN]	FY-Nodal Load [kN]	FY-Reaction [kN]	FY by Elements [kN]	FY by Elemental [kN]	FY by Body Loads [kN]	Σ(FY) [kN]
1 (A)	0.00	0.00	0.00	0.00	0.00	**0.00**	0.00	3.25	-3.25	0.00	0.00	**0.00**
2 (B)	0.00	0.00	0.00	0.00	0.00	**0.00**	0.00	0.00	0.00	0.00	0.00	**0.00**
3 (C)	0.00	0.00	0.00	0.00	0.00	**0.00**	-5.00	0.00	5.00	0.00	0.00	**0.00**
4 (D)	0.00	0.00	0.00	0.00	0.00	**0.00**	-3.00	0.00	3.00	0.00	0.00	**0.00**
5 (E)	0.00	0.00	0.00	0.00	0.00	**0.00**	0.00	4.75	-4.75	0.00	0.00	**0.00**
6 (F)	0.00	0.00	0.00	0.00	0.00	**0.00**	0.00	0.00	0.00	0.00	0.00	**0.00**
7 (G)	0.00	0.00	0.00	0.00	0.00	**0.00**	0.00	0.00	0.00	0.00	0.00	**0.00**
8 (H)	0.00	0.00	0.00	0.00	0.00	**0.00**	0.00	0.00	0.00	0.00	0.00	**0.00**

4. Elements Equilibrium



Element ID	FX at i [kN]	FX at j [kN]	Σ(FX) [kN]	FY at i [kN]	FY at j [kN]	Σ(FY) [kN]	Σ(MZ) at i [kN·m]	Σ(MZ) at j [kN·m]
1	-3.25	3.25	**0.00**	0.00	0.00	**0.00**	**0.00**	**0.00**
2	-3.25	3.25	**0.00**	0.00	0.00	**0.00**	**0.00**	**0.00**
3	-4.75	4.75	**0.00**	0.00	0.00	**0.00**	**0.00**	**0.00**
4	-4.75	4.75	**0.00**	0.00	0.00	**0.00**	**0.00**	**0.00**
5	3.25	-3.25	**0.00**	3.25	-3.25	**0.00**	**0.00**	**0.00**
6	4.33	-4.33	**0.00**	2.17	-2.17	**0.00**	**0.00**	**0.00**
7	4.33	-4.33	**0.00**	-2.17	2.17	**0.00**	**0.00**	**0.00**
8	4.75	-4.75	**0.00**	-4.75	4.75	**0.00**	**0.00**	**0.00**
9	0.00	0.00	**0.00**	0.00	0.00	**0.00**	**0.00**	**0.00**
10	0.00	0.00	**0.00**	-4.33	4.33	**0.00**	**0.00**	**0.00**
11	0.00	0.00	**0.00**	-3.00	3.00	**0.00**	**0.00**	**0.00**
12	-1.08	1.08	**0.00**	1.08	-1.08	**0.00**	**0.00**	**0.00**
13	0.42	-0.42	**0.00**	0.42	-0.42	**0.00**	**0.00**	**0.00**

(c) 2022 EngiLab PC

This page intentionally left blank.

Chapter



License Agreement

7 License Agreement

- License Agreement (Lite Edition)
- License Agreement (Pro Edition)

7.1 EULA (Lite Edition)

EngiLab Truss.2D LITE Edition

End User License Agreement (EULA)

IMPORTANT: Read the following terms carefully before installing, using and copying **EngiLab Truss.2D LITE Edition** (the "Software Product"). By installing, using and copying the Software Product you agree to accept all of the following terms. If you do not agree with the terms of this EULA, you must not use the Software Product.

This End-User License Agreement ("EULA"), effective as of the date you accept the terms hereof, is a legal agreement entered into between EngiLab PC ("EngiLab") and you (either an individual or a single entity), the end user of the Software Product identified above, which includes computer software and may include associated media, printed materials, and online or electronic documentation. The parties agree as follows:

1. SCOPE

In accordance with the terms herein, EngiLab grants to you, and you accept from EngiLab, a non-exclusive, non-transferable and non-assignable, limited license to use the current version of EngiLab's Software Product. The Software Product is protected by copyright laws and international copyright treaties, as well as other intellectual property laws and treaties. The Software Product is licensed, not sold. All rights not expressly granted are reserved by EngiLab.

2. DEFINITIONS

Software Product means the EngiLab free (Lite Edition) software accompanying this EULA, which includes executable modules and electronic documentation and may include associated media, printed materials and information available at the product web site protected by copyright laws. The Software Product also includes any updates and supplements to the original Software Product provided by EngiLab. Any software provided along with the Software Product that is associated with a separate end-user license agreement is licensed to you under the terms of that license agreement.

3. GRANT OF LICENSE

EngiLab grants you a limited, non-exclusive, non-transferable, non-renewable license to install, use, access, display, run, or otherwise interact with the Software Product. This Lite Edition of the Software Product is provided to you free of charge for your own **personal**, **academic** or **educational use only**, and **it may not be used for any commercial purposes**. For commercial use that includes all fields that do not belong to pure personal use: e.g. all kinds of use in the range of any profession or for the utilization of additional professional features, you must license the Professional (PRO) Edition of the Software Product which is subject to charging.

Besides the limitation for non-commercial use, the Lite Edition of the Software Product has also other limitations compared to the Pro Edition, on the total Number of Nodes, Materials and

Sections of a Model. The Lite Edition of the Software product can create and modify any structural Model, but it can only analyze Models with up to 10 Nodes, 3 Materials and 3 Sections.

The EULA permits use of the Software Product during unlimited period free of charge. The Software Product may be used as many times as you like, for as long as you like. You may copy the Software Product only for your own use or for your own backup purposes, provided that you keep this copyright notice and disclaimer of warranty intact. You may not copy or distribute copies of the Software Product to any third party for any use. You must not charge money or fees for the Software Product to anyone.

4. DESCRIPTION OF OTHER RIGHTS AND LIMITATIONS

4.1 You may not resell, or otherwise transfer for value, the Software Product under any circumstances. You may not charge any fees for the copy or use of the Software Product itself. You must not represent in any way that you are selling the Software Product itself.

4.2 You may not make modifications to the Software Product, or decompile, disassemble, reverse engineer or modify the Software Product or any portion of it. You may not combine other commercial applications with, or otherwise prepare derivative works of the Software.

4.3 The Software Product is licensed as a single product. Its component parts may not be separated not within the Software Product.

4.4 This EULA does not grant you any rights in connection with any trademarks or service marks of EngiLab.

4.5 You may not rent, lease, or lend the Software Product to other users.

4.6 You may not make any transfers of this EULA and Software Product to a third party.

5. COPYRIGHT

The title and all copyrights in and to the Software Product (including but not limited to any images, photographs, animations, video, audio, music, text, and "applets" incorporated into the Software Product), the accompanying printed materials, and all copies of the Software Product are owned at all the time by EngiLab. All title and intellectual property rights in and to the content which may be accessed through use of the Software Product is the property of the respective content owner and may be protected by applicable copyright or other intellectual property laws and treaties. This EULA grants you no rights to use such content. Software Product documentation is provided in electronic form. You may print copies of such electronic documentation.

6. LIMITED WARRANTY

EngiLab has made every effort to make the use of the Software Product as reliable and safe as possible. To the best of our knowledge this software is accurate and complies with the standards of good engineering practice. However, no responsibility whatsoever is accepted to any person or company whatsoever, nor is any duty or obligation owed to them as regards the accurate and safe use of this software or part thereof. EngiLab expressly disclaims any warranty that the Software Product will meet your requirements or operate under your specific conditions of use. EngiLab makes no warranty that operation of the Software Product will be secure, error free, or free from interruption. The Software Product and any related documentation is provided "as is" without warranty of any kind, either express or implied, including, without limitation, the implied warranties or merchantability, fitness for a particular purpose, or noninfringement. The entire risk arising out of use or performance of the Software Product remains with you.

(c) 2022 EngiLab PC

YOU MUST DETERMINE WHETHER THE SOFTWARE PRODUCT SUFFICIENTLY MEETS YOUR REQUIREMENTS FOR SECURITY AND UNINTERRUPTABILITY. YOU BEAR SOLE RESPONSIBILITY AND ALL LIABILITY FOR ANY LOSS INCURRED DUE TO FAILURE OF THE SOFTWARE PRODUCT TO MEET YOUR REQUIREMENTS. ENGILAB WILL NOT, UNDER ANY CIRCUMSTANCES, BE RESPONSIBLE OR LIABLE FOR THE LOSS OF DATA ON ANY COMPUTER OR INFORMATION STORAGE DEVICE.

7. DISCLAIMER OF DAMAGES

Under no circumstances shall EngiLab, its directors, officers, employees or agents be liable to you or any other party for indirect, consequential, special, incidental, punitive, or exemplary damages of any kind (including, without limitation, damages for loss of business profits, business interruption, loss of business information, or any other pecuniary loss) resulting from this EULA, or from the furnishing, performance, installation, use, or inability to use the Software Product, whether due to a breach of contract, breach of warranty, or the negligence of EngiLab or any other party, even if EngiLab is advised beforehand of the possibility of such damages and known defects. To the extent that the applicable jurisdiction limits EngiLab's ability to disclaim any implied warranties, this disclaimer shall be effective to the maximum extent permitted.

8. TERM AND TERMINATION

8.1 This EULA comes into effect when you install the Software Product on your computer, and is effective for the entire period of use of the Software Product.

8.2 Any use in violation of this EULA shall constitute not only breach of this EULA, but a violation of national and international copyright laws. Any use of the Software Product that infringes upon EngiLab's intellectual property rights or that is for commercial purposes will be investigated and EngiLab shall have the right to take appropriate civil and criminal legal action.

8.3 Without prejudice to any other rights, EngiLab may terminate this EULA if you fail to comply with the terms and conditions of this EULA. In such event, you must destroy all copies of the Software Product and all of its component parts.

7.2 EULA (Pro Edition)

EngiLab Truss.2D PRO Edition

End User License Agreement (EULA)

IMPORTANT: Read the following terms carefully before installing, using and copying **EngiLab Truss.2D PRO Edition** (the "Software Product"). By installing, using and copying the Software Product you agree to accept all of the following terms. If you do not agree with the terms of this EULA, you must not use the Software Product.

This End-User License Agreement ("EULA"), effective as of the date you accept the terms hereof, is a legal agreement entered into between EngiLab PC ("EngiLab") and you (either an individual or a single entity), the end user of the Software Product identified above, which includes computer software and may include associated media, printed materials, and online or electronic documentation. The parties agree as follows:

1. SCOPE

In accordance with the terms herein, EngiLab grants to you, and you accept from EngiLab, a non-exclusive, non-transferable and non-assignable, limited license to use the current version of

EngiLab's Software Product. The Software Product is protected by copyright laws and international copyright treaties, as well as other intellectual property laws and treaties. The Software Product is licensed, not sold. All rights not expressly granted are reserved by EngiLab.

2. DEFINITIONS

Software Product means the EngiLab paid (Pro Edition) software accompanying this EULA, which includes executable modules and electronic documentation and may include associated media, printed materials and information available at the product web site protected by copyright laws. The Software Product also includes any updates and supplements to the original Software Product provided by EngiLab. Any software provided along with the Software Product that is associated with a separate end-user license agreement is licensed to you under the terms of that license agreement.

3. GRANT OF LICENSE

EngiLab grants you a limited, non-exclusive, non-transferable, non-renewable license to install, use, access, display, run, or otherwise interact with the Software Product. This Pro Edition of the Software Product is provided to you for any legal use (**personal**, **educational**, **academic**, **non-profit** or **commercial use**).

The Pro Edition of the Software Product requires a **License Key** in order to operate. Upon purchase of the Software Product, the user automatically receives a License Key via email which is to be used with the specific Software Product. The License Key is bound to the specific user via the "Registration Name" that was given by the user and it certifies that the copy of the program is original.

The Software Product needs also an **Activation Key** in order to be activated and run. Normally, after the user enters the License Key, the program connects with EngiLab Servers in order to obtain an Activation Key online. The Activation Key is bound to the specific License Key and to the specific Computer that asked for it. Unlike the License Key that can be used in more than one Computers, an Activation Key obtained for one Computer cannot be used to activate the Software Product on another Computer.

One User - Two Machines per License Policy: The maximum number of allowed Activations per License Key is two (2). This means that a user having purchased the Software Product and being given a License Key is eligible for 2 Activation Keys, for use with two different Computers, provided only one computer is in use at any given time. The two different Computers will have the same License Key, but different Activation Keys. As a result, a user can use the same License Key for his primary (e.g. Desktop) and secondary (e.g. Laptop) Computer and each Computer can obtain a unique Activation Key bound to it.

The EULA permits use of the Software Product during unlimited period of time. The Software Product may be used as many times as you like, for as long as you like. You may copy the Software Product only for your own use or for your own backup purposes, provided that you keep this copyright notice and disclaimer of warranty intact. You may not copy or distribute copies of the Software Product to any third party for any use. You must not charge money or fees for the Software Product to anyone.

4. DESCRIPTION OF OTHER RIGHTS AND LIMITATIONS

4.1 You may not resell, or otherwise transfer for value, the Software Product under any circumstances. You may not charge any fees for the copy or use of the Software Product itself. You must not represent in any way that you are selling the Software Product itself.

(c) 2022 EngiLab PC

4.2 You may not make modifications to the Software Product, or decompile, disassemble, reverse engineer or modify the Software Product or any portion of it. You may not combine other commercial applications with, or otherwise prepare derivative works of the Software.

4.3 The Software Product is licensed as a single product. Its component parts may not be separated not within the Software Product.

4.4 This EULA does not grant you any rights in connection with any trademarks or service marks of EngiLab.

4.5 You may not rent, lease, or lend the Software Product to other users.

4.6 You may not make any transfers of this EULA and Software Product to a third party.

5. COPYRIGHT

The title and all copyrights in and to the Software Product (including but not limited to any images, photographs, animations, video, audio, music, text, and "applets" incorporated into the Software Product), the accompanying printed materials, and all copies of the Software Product are owned at all the time by EngiLab. All title and intellectual property rights in and to the content which may be accessed through use of the Software Product is the property of the respective content owner and may be protected by applicable copyright or other intellectual property laws and treaties. This EULA grants you no rights to use such content. Software Product documentation is provided in electronic form. You may print copies of such electronic documentation.

6. LIMITED WARRANTY

EngiLab has made every effort to make the use of the Software Product as reliable and safe as possible. To the best of our knowledge this software is accurate and complies with the standards of good engineering practice. However, no responsibility whatsoever is accepted to any person or company whatsoever, nor is any duty or obligation owed to them as regards the accurate and safe use of this software or part thereof. EngiLab expressly disclaims any warranty that the Software Product will meet your requirements or operate under your specific conditions of use. EngiLab makes no warranty that operation of the Software Product will be secure, error free, or free from interruption. The Software Product and any related documentation is provided "as is" without warranty of any kind, either express or implied, including, without limitation, the implied warranties or merchantability, fitness for a particular purpose, or noninfringement. The entire risk arising out of use or performance of the Software Product remains with you.

YOU MUST DETERMINE WHETHER THE SOFTWARE PRODUCT SUFFICIENTLY MEETS YOUR REQUIREMENTS FOR SECURITY AND UNINTERRUPTABILITY. YOU BEAR SOLE RESPONSIBILITY AND ALL LIABILITY FOR ANY LOSS INCURRED DUE TO FAILURE OF THE SOFTWARE PRODUCT TO MEET YOUR REQUIREMENTS. ENGILAB WILL NOT, UNDER ANY CIRCUMSTANCES, BE RESPONSIBLE OR LIABLE FOR THE LOSS OF DATA ON ANY COMPUTER OR INFORMATION STORAGE DEVICE.

7. DISCLAIMER OF DAMAGES

Under no circumstances shall EngiLab, its directors, officers, employees or agents be liable to you or any other party for indirect, consequential, special, incidental, punitive, or exemplary damages of any kind (including, without limitation, damages for loss of business profits, business interruption, loss of business information, or any other pecuniary loss) resulting from this EULA, or from the furnishing, performance, installation, use, or inability to use the Software Product, whether due to a breach of contract, breach of warranty, or the negligence of EngiLab or any other party, even if

EngiLab is advised beforehand of the possibility of such damages and known defects. To the extent that the applicable jurisdiction limits EngiLab's ability to disclaim any implied warranties, this disclaimer shall be effective to the maximum extent permitted.

8. TERM AND TERMINATION

8.1 This EULA comes into effect when you install the Software Product on your computer, and is effective for the entire period of use of the Software Product.

8.2 Any use in violation of this EULA shall constitute not only breach of this EULA, but a violation of national and international copyright laws. Any use of the Software Product that infringes upon EngiLab's intellectual property rights will be investigated and EngiLab shall have the right to take appropriate civil and criminal legal action.

8.3 Without prejudice to any other rights, EngiLab may terminate this EULA if you fail to comply with the terms and conditions of this EULA. In such event, you must destroy all copies of the Software Product and all of its component parts.

(c) 2022 EngiLab PC



www.engilab.com